State of Florida

Department of Transportation

FDOT MicroStation Essentials – Part I

CE-11-0115

Course Guide
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PRODUCTION SUPPORT CADD OFFICE
TALLAHASSEE, FLORIDA

http://www.fdot.gov/cadd/

FDOT MicroStation Essentials – Part I
CE-11-0114

Description

For the new MicroStation user, this hands-on course builds a solid foundation on the concepts of MicroStation and the use of its tools and features for creating and editing designs. Beginning with setting up a drawing and concluding with complex elements, the participant is walked through a typical workflow using the tools and features of MicroStation. Interspersed throughout are real-world design problems solved using the concepts learned up to that point in the course. Various engineering disciplines are represented in the form of hands-on exercises.

The course will provide a good user understanding of basic MicroStation with bases set in the Florida Department of Transportation (FDOT) workspace.

Topics Covered

This course includes but is not limited to:

- Design Files
- Design Environment
- Viewing and Zooming
- Models
- Levels
- Basic Drawing Tools
- Drawing with Precision
- Changing Elements
- Selecting and Grouping

Prerequisites

No prior CADD experience is required, but some manual drafting or related CADD product experience is recommended.

Duration: *16 Hours*

Professional Credit Hours: *NONE*

1 DESIGN FILES

INTRODUCTION

This course was developed to introduce MicroStation users to the V8i SELECTseries4~~2~~ Essentials tools for the design and development of Florida Department of Transportation (FDOT) projects. The intent is to cover the basics and give the user general knowledge of MicroStation with FDOT's customization to assist the user in producing a product that is in compliance with the FDOT Computer Aided Design and Drafting (CADD) Standards.

OBJECTIVES

This chapter reviews the MicroStation design environment within the Florida Department of Transportation (FDOT) Workspace. Special consideration is given to:

- MicroStation Design Files

- FDOT Seed Files

- Launching MicroStation in FDOT Workspace

- File Open in FDOT Workspace

- FDOT Discipline Configurations

- Creating Files in FDOT Workspace

- Design File Settings

- FDOT Preferences

- MicroStation Help and Support

- FDOT Help and Support

MICROSTATION DESIGN FILES

The MicroStation V8i format is a subset of the .DGN file format used by previous versions of MicroStation, and the .DWG file format used by AutoCAD. This gives MicroStation native support for both products, as well as the .DXF format.

A design file (.DGN), often referred to as design herein, is a container for smaller containers called models. Each model consists of graphical elements such as arcs, lines and circles. Each model is a unique drawing within the design file, which allows multiple drawings to reside in the same design file. For example, a roadway design, profiles, and plan and profile sheets can exist as separate models in a single design file.

Note   Only one design file is opened at a time in MicroStation, but other design files may be viewed by attaching them as References to the active model in the open design file.
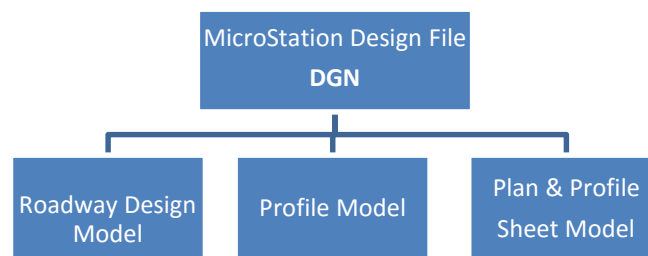
FDOT SEED FILES

Each MicroStation design file is created form a seed file. A seed file is a previously stored design file, with user specified settings, used as a template to create a new DGN. The Florida Department of Transportation (FDOT) has created seed files for use in production that have standard settings stored within them. Available seed files for the FDOT's purposes are:

FDOTSEED2D.DGN	Contains settings to create a 2D design file.
FDOTSEED3D.DGN	Contains settings to create a 3D design file.
FDOTSEED2D.DWG	Contains settings to create AutoCAD files.
FDOTSEEDKEYMAP.DGN	A 2D design file containing additional models setup for Key Maps.
FDOTSEEDXS.DGN	A 2D design file containing additional models setup for Cross Sections.
RWSEED2D.DGN	A 2D design file containing unit labels specific to Right of Way.
STRUCTURESSEED.DGN	A 2D design file containing unit labels & settings specific to Structures.
STRUCTURESSEED3D.DGN	A 3D design file containing unit labels & settings specific to Structures.

LAUNCHING MICROSTATION IN FDOT WORKSPACE

FDOT has created and provided a custom workspace to standardize the MicroStation environment for agency use. This workspace determines how MicroStation displays on the screen and sets up certain default files and search paths. FDOT's customized MicroStation workspace can be launched by the following common avenues:

Double-click the FDOT icon from the FDOT desktop folder.

From the Windows Start menu, select Start > All Programs > FDOTSS4 > FDOTSS4.

Drag and Drop any design file with a DGN extension onto the FDOTSS4 icon in the FDOT desktop folder. A MicroStation session with the FDOT workspace will launch with the selected file.
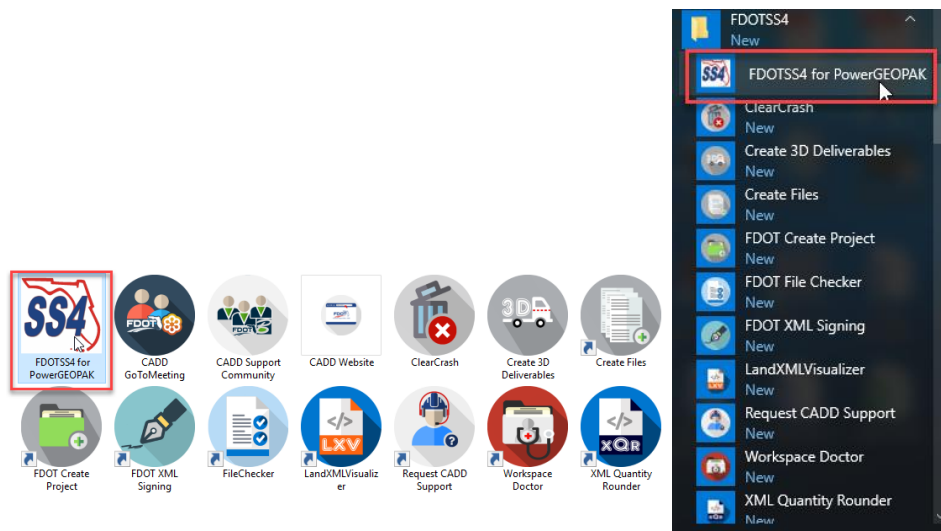
Note   Opening a MicroStation session from the Bentley program group or by double clicking on a DGN file will open MicroStation in an Untitled workspace and not within the FDOT workspace.

FILE OPEN IN FDOT WORKSPACE

When MicroStation launches without a specified a design file, the File Open dialog will display. File Open is used to manage design files, performing file-related tasks, such as creating and opening drawings.
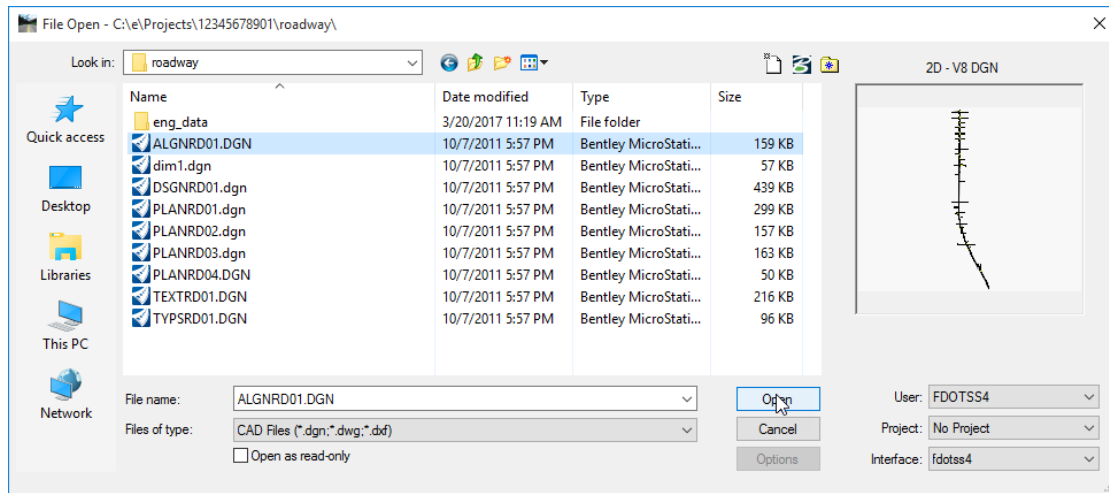
Note   The "Classic" Open dialog is set to display through the MicroStation menu option Workspace > Preferences > Look and Feel by "un-checking" the Use Windows File Open Dialog checkbox.

The File Open dialog contains standard navigation tools such as a drive list and files list. Selecting a file from the central window displays a preview of the selected file on the right-hand side. Right clicking on a file opens a drop down menu with file options for use on the selected file.
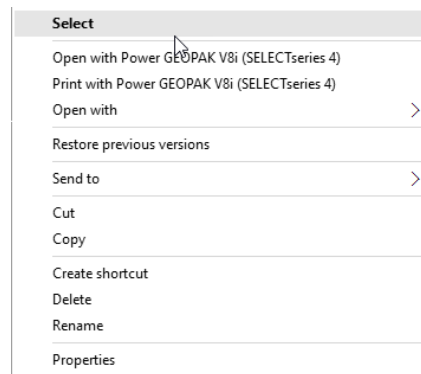
The File name identifies a specific file selection and List Files of Type filters which type of files display in the Files list box. For example, to view DGN files only, change the option to MicroStation DGN Files [*.dgn].

The Options button only displays when selecting DWG file formats. FDOT Software delivery provides preset settings found within this option and does not recommend users to make any changes.

FDOT WORKSPACE COMPONENTS

A workspace in MicroStation is a collection of predefined settings and configuration variables that accommodate agency standards and configurations for their work environment. FDOT has created its own workspace to accommodate FDOT CADD Standards.

Project configuration files exist in order to facilitate sharing of configurations among users in a workgroup. The settings in each *User* configuration file allow each user in a workgroup to customize MicroStation, while the shared settings of the *Project* configuration file provide uniformity within the workgroup. The project configuration file is the file that is selected when the *Project* component is set in the *Workspace* section in MicroStation Manager. The FDOT created workspace is composed of 3 areas:
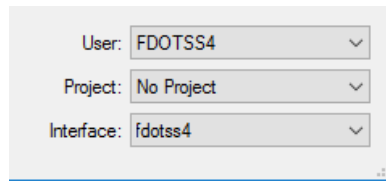
- **User** - Sets *User* configuration variables. This component in effect points to the active workspace's Project and Interface components. The user name identifies the user configuration file (.ucf), which is stored in ***FDOTSS4 \ Workspace \ Users*** directory. This should default to **FDOTSS4**.
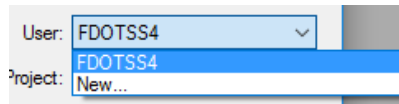
- **Project** - Sets Project configuration variables specific to a certain project, such as path location of files. A project consists of customized data files identified by configuration variables in the project configuration file (.pcf), which is stored in the local ***E \ Projects*** directory created for FDOTSS4. The dropdown list views all Projects created with a .pcf file and defaults to the last Project opened. If no projects have been previously created then this will default to **No Project**.
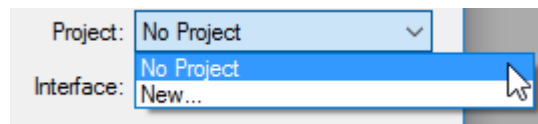
- **Interface** - Sets the active user interface. FDOT has a customized user interface defined in a DGN library that is stored in the local FDOTSS4 \ Workspace \ Interfaces\ MicroStation directory. This should default to **FDOTSS4**.
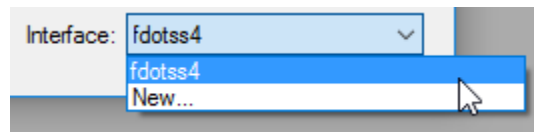
> *Note*   FDOTSS4 has customized (User) Preferences defined in a user preference file (.upf) that is stored in the local FDOTSS4 \ Workspace \ Prefs directory. This file is updated when the user makes any preference selections and "Saves Settings".

SWITCHING DESIGN FILES

Once a DGN file is open, users are allowed to switch to other DGN files by selecting the MicroStation menu option **File > Open** <OR> hold the Control key down and select the letter 'O' (Ctrl+O)**.** The Open dialog displays with abbreviated options as in the File Open dialog. It is not necessary to return to the MicroStation File Open dialog to open another drawing.
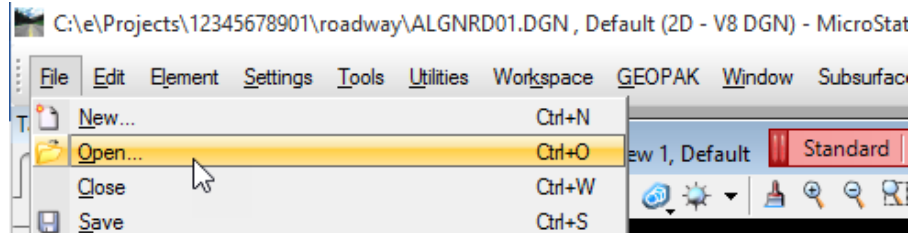
 To navigate to a different directory, select the directory form the *Directories* list in the middle of the dialog. To open a design file, select it from the *Files* list and click OK, <OR> double-click on the file name to open it.
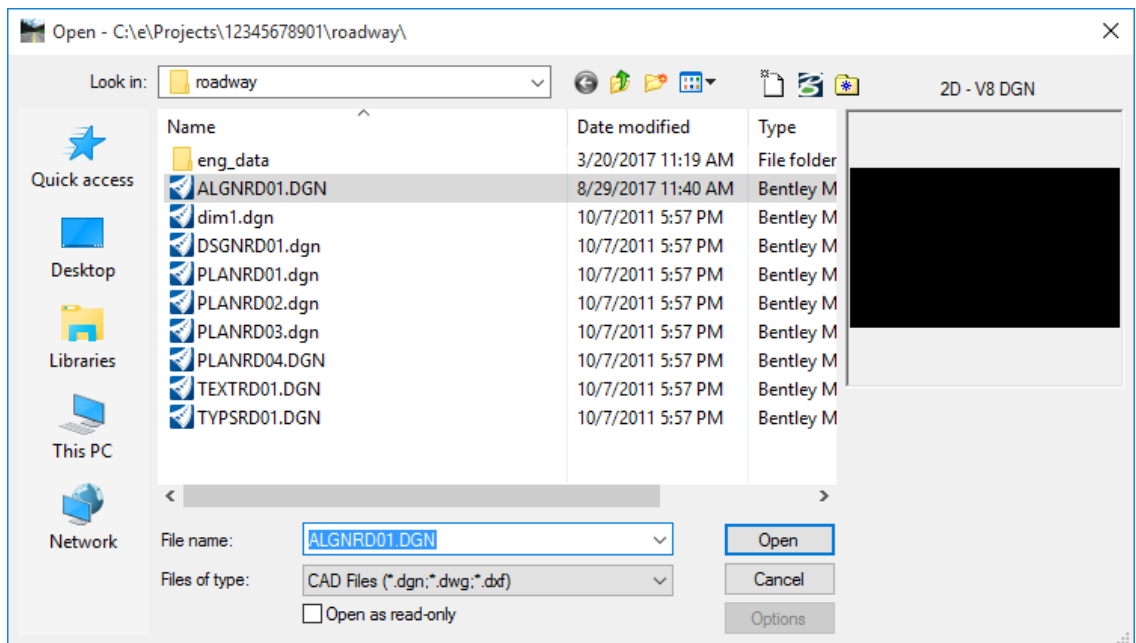
FDOT DISCIPLINE CONFIGURATIONS

FDOT CADD Software is delivered with Configurations within the FDOT Workspace which dictate discipline specific FDOT Menu options, file specification and resource files for each of the FDOT Standards for Roadway, Structures, Right of Way, Utilities and Photogrammetry Disciplines. Users may change to the appropriate Configuration while in a **MicroStation** session from the FDOT Menu option: Standard > Configuration.

The following reflects just the FDOT Standard menu bar which discipline specific menus build upon. Further details regarding the FDOT Menu options are covered in Chapter 2 of this document.

Any design file can be opened in any configuration, but the appropriate Configuration must be used with the associated discipline design files to gain access to the corresponding tools and resource files used within the FDOT automated tools and utilities.
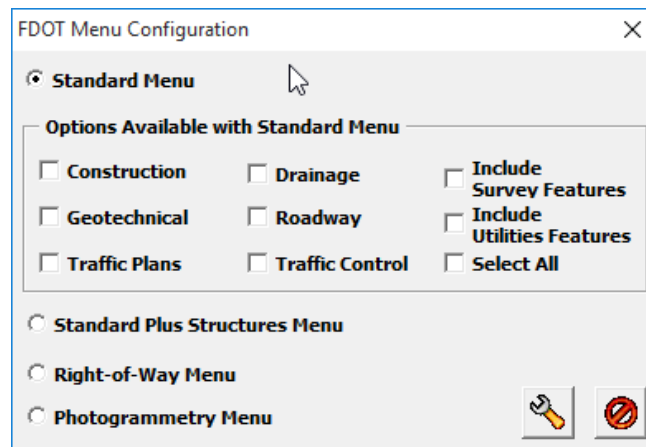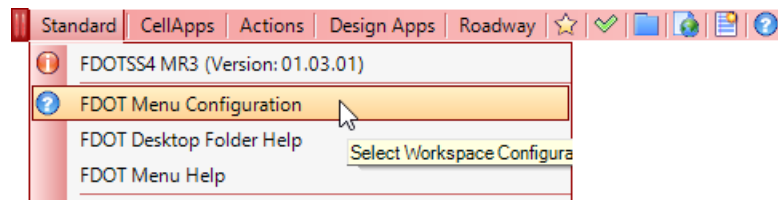
Roadway Menu (Only three of the available Standard Menu Discipline Options display below):
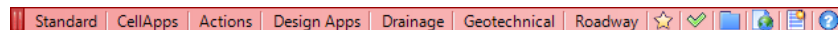
Structures Menu:
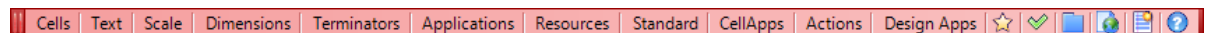
Right of Way Menu:
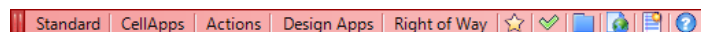
Photogrammetry Menu:
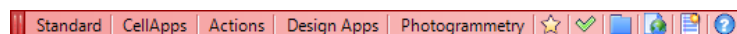
CREATING FILES IN FDOT WORKSPACE

While standard MicroStation tools can be used to create new files, FDOT provides a utility to create new files with the correct seed file and file naming structure already in place. This FDOT utility can be used within a MicroStation session or outside of MicroStation.

To access this FDOT utility within a MicroStation session, select the FDOT Menu option **Actions > Create/Edit File**:
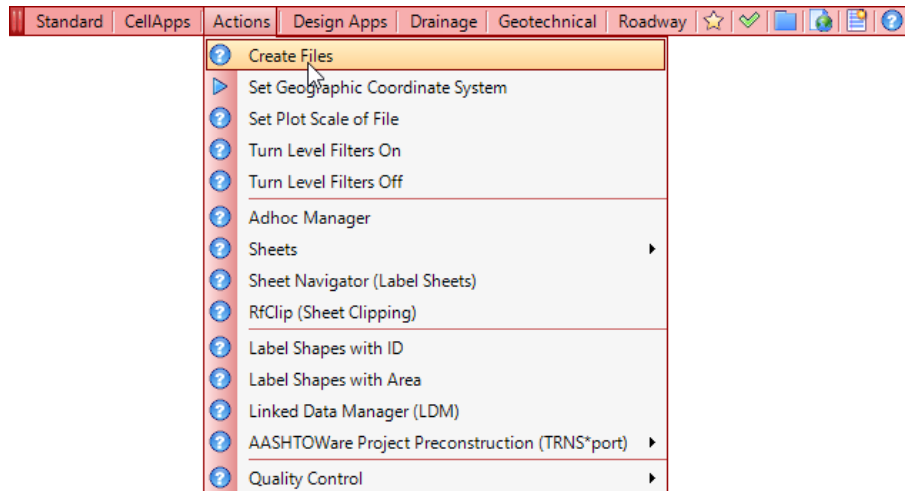
<OR> to access from outside MicroStation, double-click the **Create Files** icon from the FDOT desktop folder.
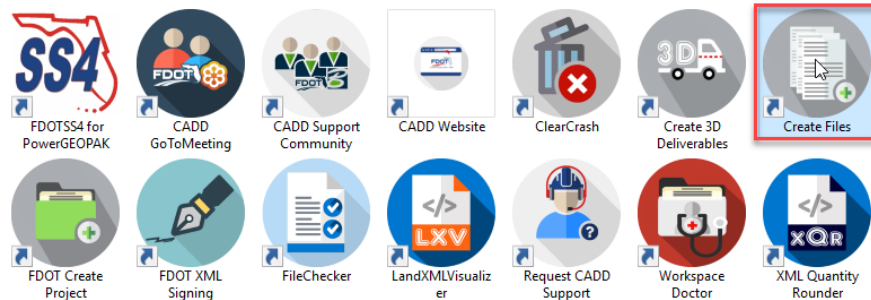
This opens the Create File/Project dialog.

CREATE FILE / PROJECT DIALOG

The Create File/Project application is used to create MicroStation design files, Compbook Excel files, and other files in accordance with the FDOT standard file naming conventions. Create File/Project uses an ASCII text file, called a Control File (*.ctl) to perform these task(s).

Note   This application can also create projects, although Electronic Delivery is the recommended method for creating new FDOT projects.

A description of the Create File/Project dialog is listed below:

Project   Defines the file path where files for the project will be created. Clicking the **Project Settings** button allows for the creation of a new project <OR> navigation to select an existing project to use.

Workspace   Defines the current workspace.

Control File   Defines the control file, which sets all other information in Create File/Project dialog. Use the *Control File* dropdown arrow to select the appropriate control file. There are five available control files:

        GEOTECH.CTL for Geotechnical related files.

        MECHELEC.CTL for Mechanical and Electrical related files.

        ROADWAY.CTL for Roadway related files.

        RW.CTL for Right-of-Way related files.

        STRUCTURES.CTL for Structures related files.

        Upon selection of a *Control File*, all other fields automatically populate with FDOT standard information.

File Group   Each control file has several *File Groups* to choose from. Each group is a specific area of design. Selecting a file group automatically loads the file types related to that group into the *File Type* list box.

File Type   Selecting a *File Type* automatically sets all of the settings for the *Output File (Name), Output folder,* and *Seed File* fields, of the file to be created. If a file with the same name already exists, the numbering is automatically incremented.

Output File

Output Folder   These settings are automatically set when the *Control File*, *File Group* and *File Type* are specified. Only the *Output File* may be modified directly. **Browse** buttons are provided to change the *Output Folder* or the *Seed Path*, if needed.

Seed File

Seed Path

Action   When a MicroStation design file is created and opened, this text field can be used to enter MicroStation commands, such as **set active scale** to 10 (AS=10). To execute the command, click the **Apply Action** button. Action functions similarly to the MicroStation Key-in utility.

Create   Selecting the **Create** button executes the creation of a new DGN file as specified by the selections made and saves the file to the *Output File* and *Output Folder* specified. Click the *Create* button more than once to create multiple new DGN files where needed. The application will automatically increment the numbering of subsequent files with the same file name. This is useful when more than one blank copy of a particular type of DGN file is needed, such as, Plan and Profile Sheets.

Open File   Selecting the **Open File** button opens the newly created MicroStation file.

➢ *To create a new design file:*

1. Open the Create File/Project dialog by selecting FDOT Menu option: **Actions > Create/Edit File**. The Create File/Project dialog displays.

2. Click the **Project Settings** button, and then click the **Select Active Project** button to select an existing Active Project.

Note   Create a New Project is covered in the Plans Preparation training course.
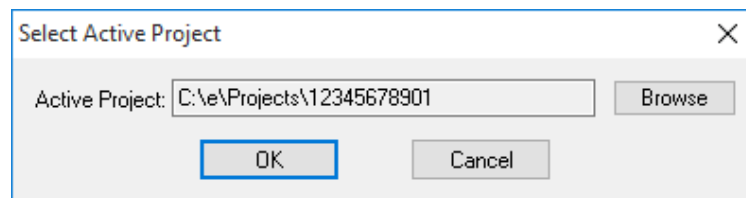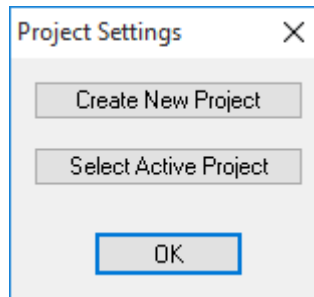
3. From the Select Active Project dialog, click **OK** and then click **OK** to close the Project Settings dialog. The Create File/Project dialog displays.

4. From the Create File/Project dialog, click on the *Control File* dropdown arrow to select appropriate **Control File**:
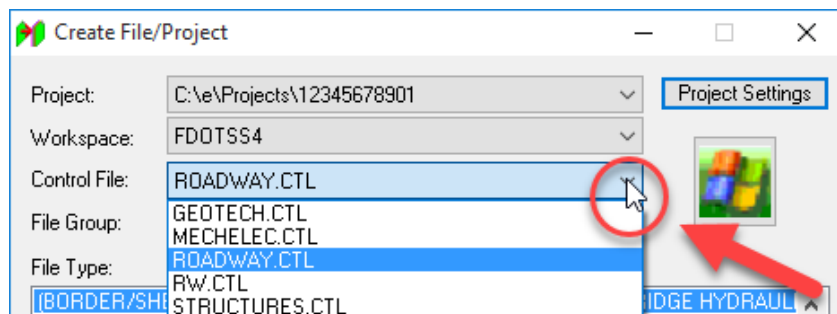
5. Click on the *File Group* dropdown arrow to select the applicable **Discipline Group**. This will populate the *File Type* window with the associated files for selection.

6. Select the **File Type** to create and then click the **Create** button. The new design file is automatically created as specified and saved under the corresponding discipline folder. The Create File/Project dialog remains open to continue creating as many new design files as needed.

Note   If creating a new Project, the FDOT standard project directory structure will be automatically created and saved where specified.

7. Select the **Open File** button to open the newly created design file in MicroStation.

DESIGN (DGN) FILE SETTINGS

When a new design file is created, it is recommended to review the set-up of the design file parameters that control how the drawing functions. The most common file settings are found in the design File Settings dialog which provides control over such settings as highlight colors, coordinate readout and grid settings. To access design File Settings, select the MicroStation menu option Settings > Design File.
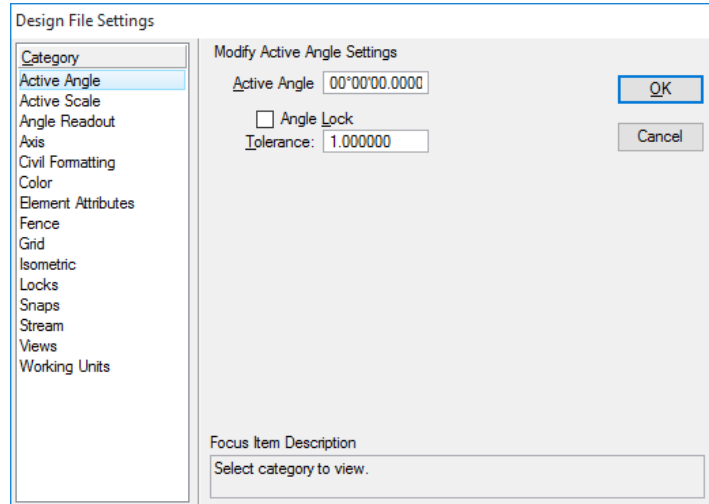
Note   It is important to note that, if the proper Seed File is used when the MicroStation file is created, then there should be NO need to change any of the design file settings.

WORKING UNITS

Working Units consists of controls that are used to set "real world" units of measurement for design models. MicroStation recognizes Metric and English units, either of which may be selected. Additionally, users can create their own custom units, by relating them to the standard units (Metric or English). Changing between the units used in a model makes no difference to the size of geometry in the model; alternatively, changing the Resolution setting in the Advanced Settings does change the size of existing geometry in the model. In practice, the Resolution setting will rarely, if ever, have to be changed from the default.
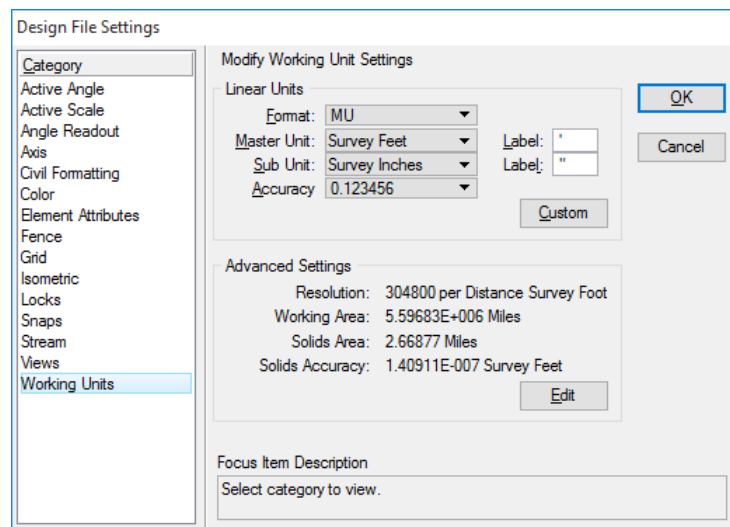
Note   Different models, contained in the same design file can have different Working Units assigned to them.

GLOBAL ORIGIN

The global origin for FDOT's standard seed files are set to an XY coordinate of 0, 0. The global origin can be relocated to specific coordinate values to create a custom coordinate system.

When using Create/File Project to create MicroStation files, a predefined seed file is used to create each design file automatically and thus the global origin is set to FDOT standards.

FDOT PREFERENCES

COMPRESSING DESIGN FILE

When elements are deleted from a MicroStation design file, the elements no longer display in the design, but a record of the deleted elements remains in the design file. The *Compress Design File* command removes the records, which reduces the file's size.

The undo buffer is cleared when a design file is compressed, so after compressing, the user will no longer be able to undo any previous operations.

It is best to use the *Compress Design File* command at the end of an editing session. This can also be set to happen automatically upon exiting the design file. To enable this setting, select **Workspace > Preferences** from the MicroStation menu bar, select the *Operation Category* and populate the checkbox for *Compress File on Exit*.
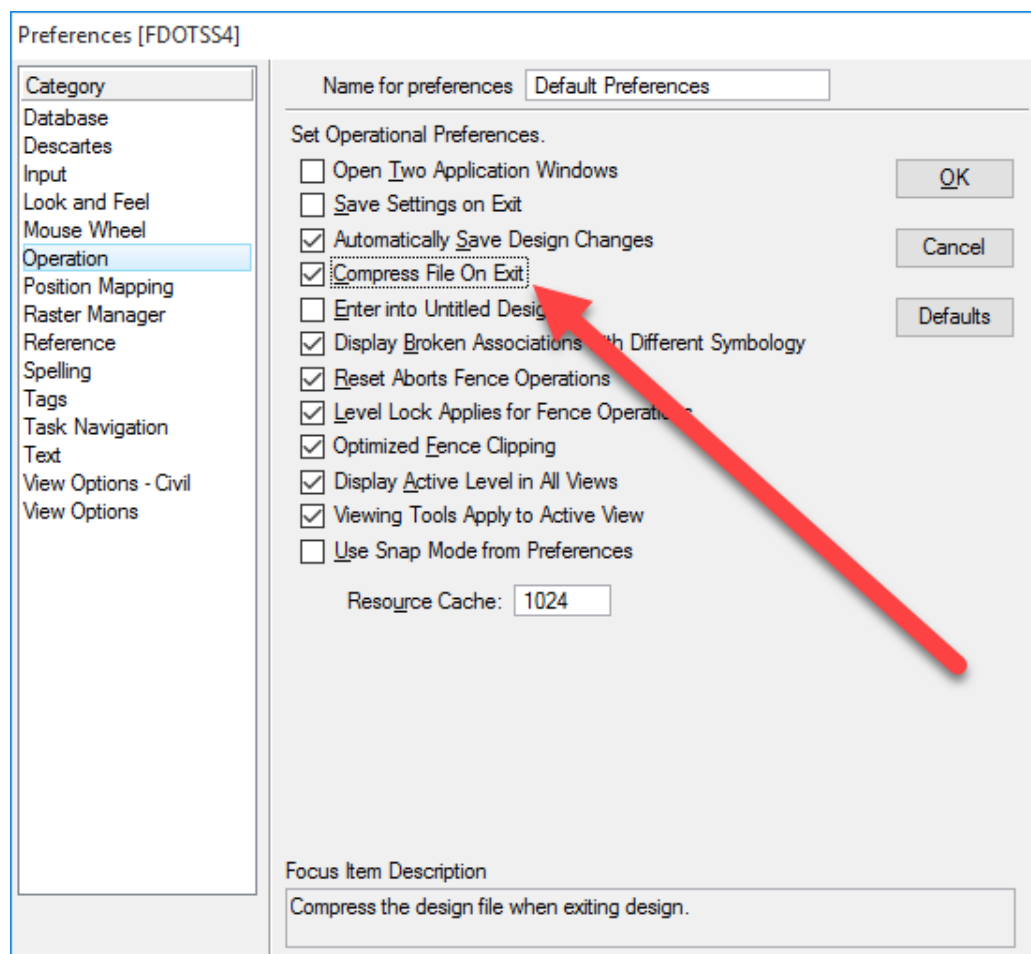
SAVING DRAWINGS

By default, MicroStation saves drawing changes automatically. MicroStation design files are saved in DGN format while AutoCAD drawing files are saved in DWG format.

The fact that the file is saved automatically is a very powerful feature of the software. This gives the user security of knowing that in the event of a computer crash or power failure, the design changes for the last completed command are saved.

Only changes to design elements are saved automatically. File settings (active color, view setup) must be saved using the MicroStation menu option: **File > Save Settings** <OR> **[Ctrl] + [F]** on the keyboard. This preserves all file settings to ensure reflection of current settings in future openings of the file.

Note Right-clicking on the FDOT Menu displays a popup menu with a Save Settings option. This is NOT the same function as described above, but only saves the location of the FDOT Menu when repositioned by the user.

To disable this feature, select the MicroStation menu option: Workspace > Preferences and then select the Operation Category in the Preferences dialog. Uncheck the Automatically Save Design Changes checkbox.
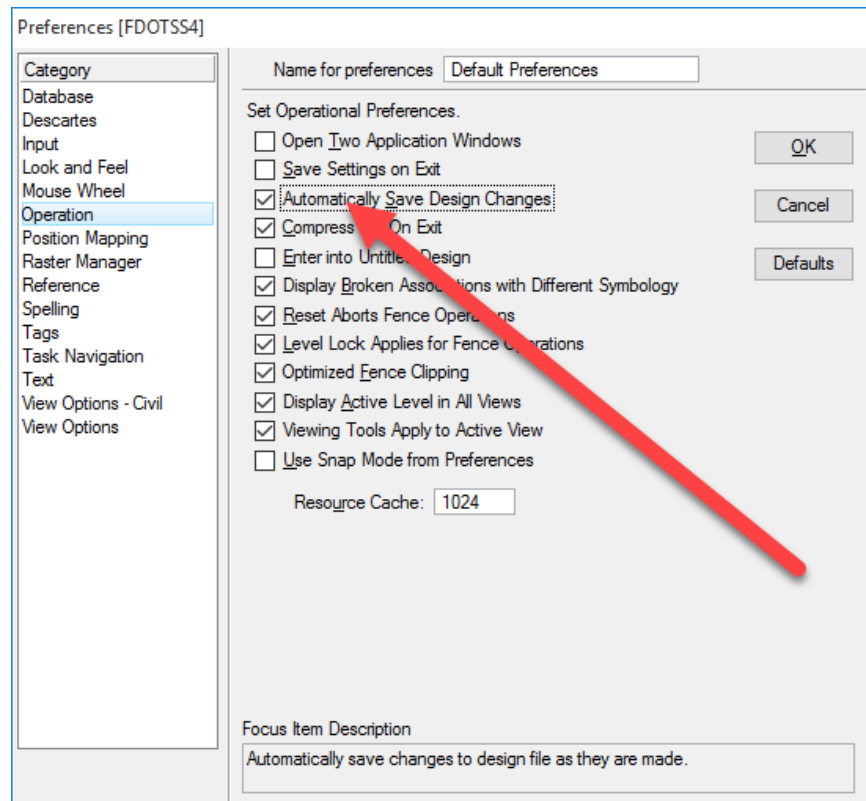
If Automatically Save Design Changes is disabled, the user must manually save the file by clicking the MicroStation menu option: File > Save. Selecting File > Save As allows the user to save the file as a V8, V7, DWG, or DXF file format. The Save As command can also be used to create a backup of the design file.

Note Save Settings can be set to execute automatically from this Operation Category of the Preferences dialog, by checking the Save Settings on Exit checkbox.

To return to the File Open dialog during an editing session, select the MicroStation menu option: **File > Close.**

MICROSTATION HELP AND SUPPORT

MicroStation provides **Help** and product support thru online services, documentation and Tool Tips that can be accessed from the MicroStation menu bar once a session is opened.
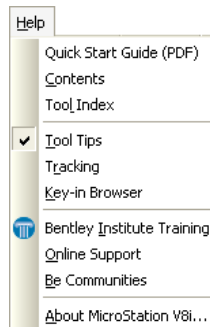
CONTENTS

The on-line help file is accessed under the *Contents* tab on the Help menu. The user can find solutions to frequently asked questions or search for help on a specific topic.
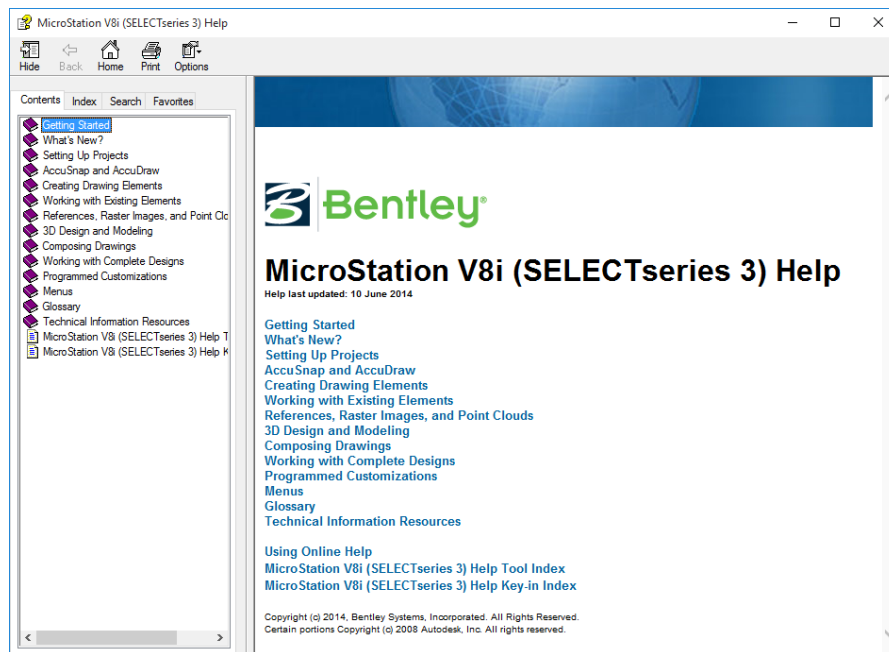
If the Index tab is selected, then an index is presented in the left window pane where the user may sort and select topics of interest. Once selected, the right window pane is populated with information on the selected topic.

FDOT HELP AND SUPPORT

The FDOT Engineering CADD Systems Office (ECSO) provides many venues of help and support whether one-on-one; self help; or group assistance. The FDOT Customer Support Guide is published on the ECSO website under the Publications link to explain how to get help, who to contact and what to expect.

From the FDOT Desktop folder, the user will find links to join a GoToMeeting for one-on-one support; connect to FDOT's forum on the Bentley Communities website; connect to the FDOT PSO|CADD's website for on-line support; and an option to submit their own Support Request via FDOT PSO|CADD's online support portal.

From within a MicroStation session, the user will find many Help and Support links under the FDOT Menu option: *Standard* as shown below.

- Throughout the FDOT Menu, Help icons, such as , can be found next to most menu options which will link the user to Help files specific to those options.

- ~~The Standards menu displays Shortcut Key ins that allows the user to execute a menu option without opening the menu and selecting the option. The use need only to be in an active MicroStation view and type: \ *<key-in>* (ex: \forum)~~

- ECSO has many links to contact Support staff for assistance or find resource material within the ECSO website.

- There is an FDOT CADD Support Forum for users to globally submit issues for any CADD user to respond. This forum is maintained and monitored by the PSO|CADD staff , but also opens the door to many other resources for assistance.

Through the ECSO's website, the user will have access to much more Help and Support assistance through Publications; Downloads; Training Courses; FDOT Training Manuals; GoTo Webinars; Quick Clips; Frequently Asked Questions (FAQ); Quick Links and links to other offices and outside professional resources.

2 DESIGN ENVIRONMENT

OBJECTIVES

This chapter discusses the design environment within MicroStation. Highlights of this chapter include:

- DGN / DWG Work Modes

- Workspace Preferences

- Screen Layout

- Standard FDOT Menu

- MicroStation Drawing Tools

INTRODUCTION

MicroStation provides an interface that is user friendly and customizable. Most tools are accessed from toolboxes. In addition to various tools, MicroStation also works in one of three modes; DGN, DWG or V7. The various modes maintain uniformity when working on older files or files in DWG format.

Additional controls for the actual design environment can be accessed via Workspace > Preferences from the MicroStation menu. The options and settings located here control several aspects of the design environment including mouse operation, look and feel and view options. These settings can be stored as defaults so that every session of MicroStation operates in a standard manner.

Drawing attributes are easy to control with MicroStation. The main controlling application is the *Attributes* tool bar, which facilitates setting the element symbology with which elements are placed into the design files.

DGN / DWG WORK MODES

MicroStation natively reads and writes both the MicroStation DGN file format and DWG file format, thus enabling less data loss from sharing. To ensure compatibility with AutoCAD DWG and MicroStation V7 files, MicroStation V8 allows users to work in three different environments; DGN Capacity Work Mode, DWG Restriction Work Mode, and V7 Work Mode.

DGN Work Mode is the native format and is indicated with a blank field in the status bar. The Work Mode changes automatically according to which file type is opened and is indicated in the lower right of the status bar by an icon.
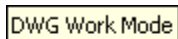
DWG Work Mode is a working environment within MicroStation specifically adapted to edit and review DWG files. DWG files can also be directly referenced from a MicroStation DGN file. When a DWG file is opened in MicroStation, various MicroStation tools are restricted to ensure that no DWG-incompatible features are added to the DWG file while editing it in MicroStation. To create DGN files that are DWG compatible, turn on DWG Work Mode. In both cases MicroStation restricts its tools to create only elements that are compatible with DWG format.

WORKSPACE PREFERENCES

The Preferences dialog contains settings that customize the way MicroStation operates and looks. To access the dialog, select the MicroStation menu option: **Workspace > Preferences**.
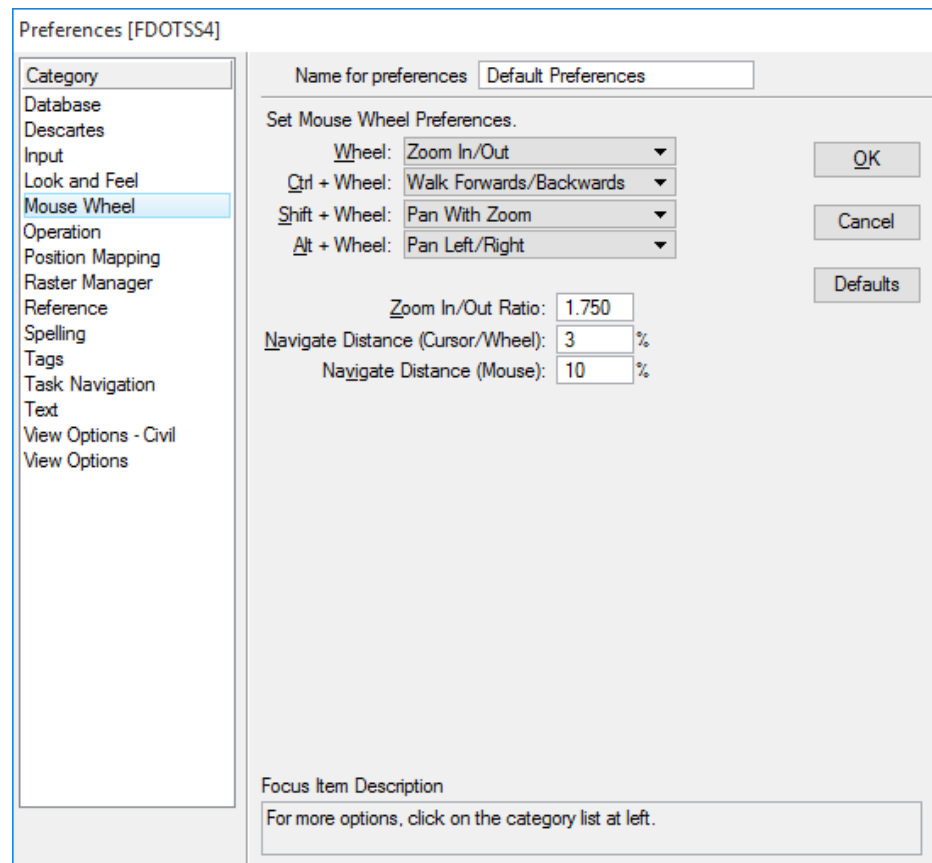
The settings in this dialog work at the system level, which means that they are not specific to any design file, but are active no matter which design file is being worked in. This dialog has many options which are referred to throughout this course manual.

MOUSE BUTTON FUNCTIONS

When using a mouse with MicroStation, the mouse buttons are used for three primary functions: **Data**, **Reset**, and **Tentative**. The Data button can also be called the pick button because it is used to "pick" tools and points on the screen. The Reset button cancels or restarts a command. The Tentative buttons snaps to points on existing elements.
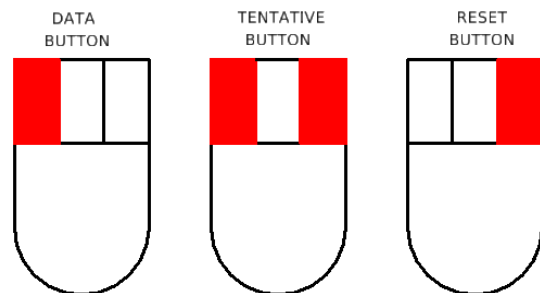
By default, the middle button is not set to Tentative, but instead Tentative is invoked by a two button chord; (clicking the left and right buttons at the same time). To change the default button assignments, select the **MicroStation** menu option: **Workspace > Button Assignments** to display the Button Assignments dialog.
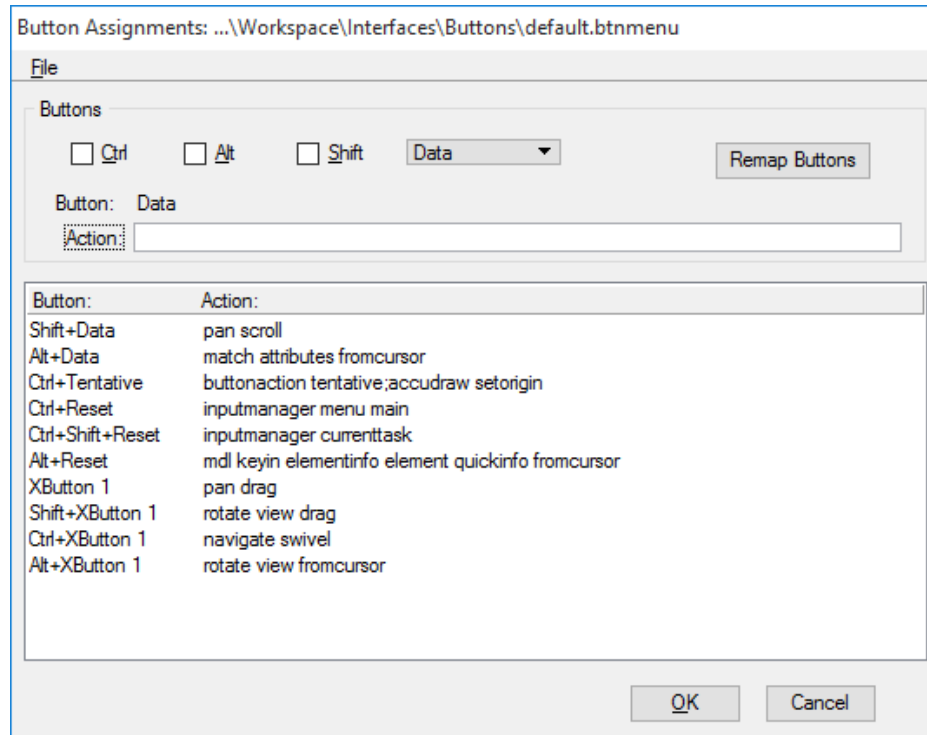
> ➢ *To Change Button Assignments:*

1. Click **Remap Buttons** button.
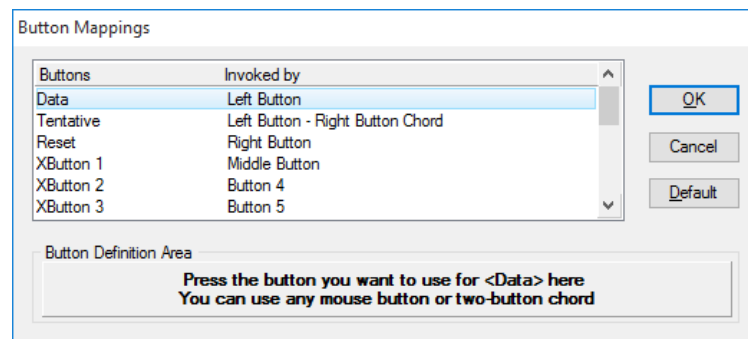
2. Select a Buttons function to assign a selected mouse button.

3. Complete instructions displayed in the *Button Definition Area*.

4. Click **OK** twice to exit

Note   A two button mouse generally uses the two button chord to invoke a tentative point.

MOUSE WHEEL FUNCTIONS

The *Mouse Wheel* can be used to pan and zoom in a drawing. Different zoom functions can be assigned to the wheel alone, using the wheel while holding the **Ctrl** key <OR> using the wheel while using the **Shift** key. Users can change the way that the wheel works through the *Mouse Wheel* category of the Preferences dialog.   To access the Preferences dialog, select the MicroStation menu option: **Workspace > Preferences**.
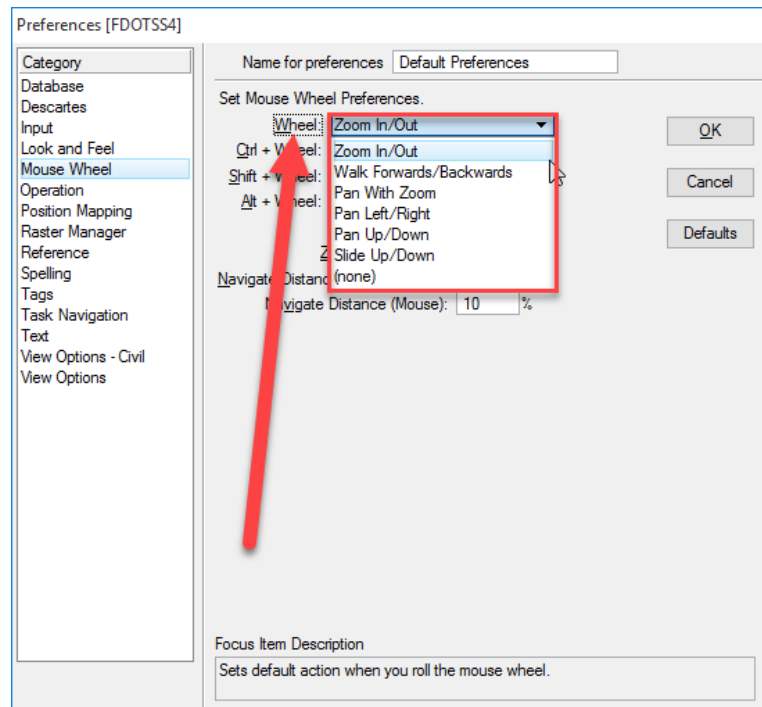
Wheel	Sets the mouse wheel to zoom in and out by default. Options are Zoom In/Out, Walk Forwards/Backwards, Pan With Zoom, Pan Left/Right, Pan Up/Down, Slide Up/Down, or (none).
Ctrl + Wheel	Sets the mouse wheel to walk forward and backward (3D only) by default. Options are Zoom In/Out, Walk Forwards/Backwards, Pan With Zoom, Pan Left/Right, Pan Up/Down, Slide Up/Down, or (none).
Shift + Wheel	Sets the mouse wheel to zoom and re-center by default. Options are Zoom In/Out, Walk Forwards/Backwards, Pan With Zoom, Pan Left/Right, Pan Up/Down, Slide Up/Down, or (none).
Alt + Wheel	Sets the mouse wheel to change the Z elevation (3D only). Options are Zoom In/Out, Walk Forwards/Backwards, Pan With Zoom, Pan Left/Right, Pan Up/Down, Slide Up/Down, or (none).
Zoom In/Out Ratio	Sets the zoom ratio increment for each turn of the mouse wheel. Default is 1.750.
Navigate Distance (Cursor/Wheel)	Obsolete, no longer supported.
Navigate Distance (Mouse)	Obsolete, no longer supported.

SCREEN LAYOUT

MicroStation's screen layout may be customized to open with components in the same position for every editing session. It may also be reorganized or changed on the fly during an editing session. FDOT has established a default layout that is set by the predefined FDOT workspace.

The following is a review of the different components that make up the screen layout.
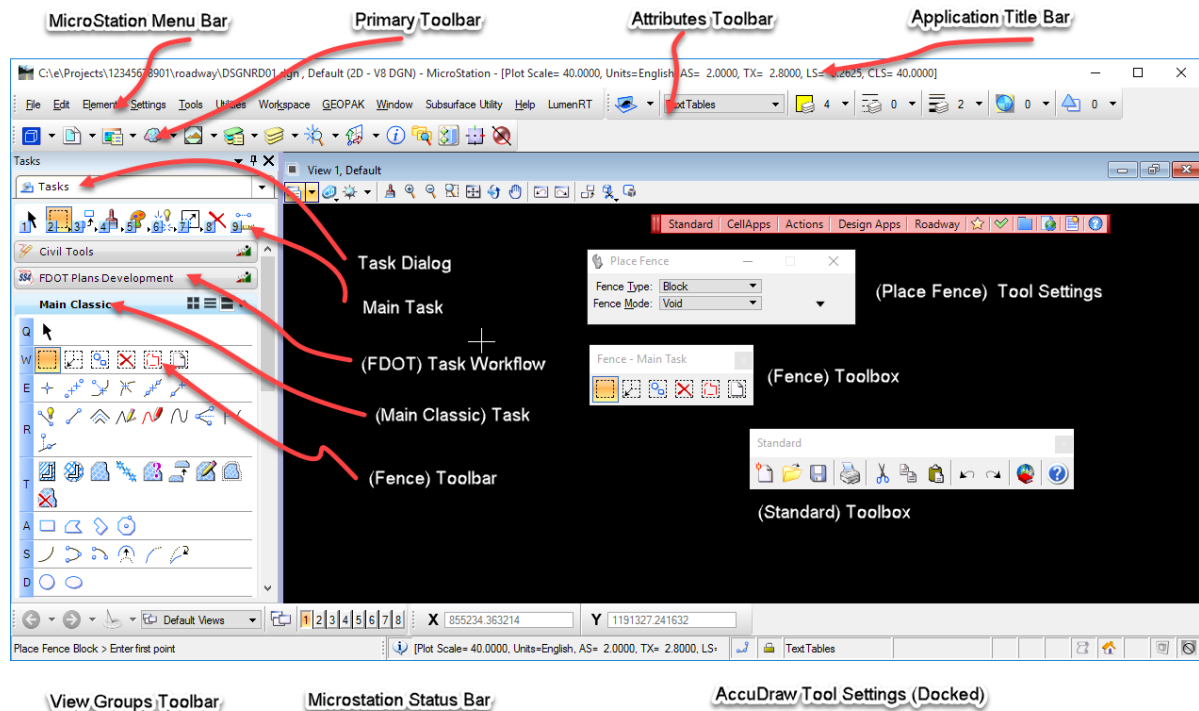
MICROSTATION MENU BAR
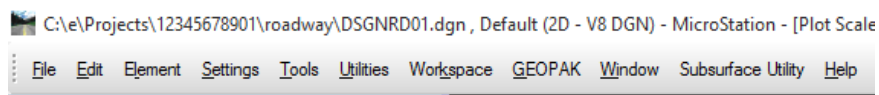
The MicroStation menu bar is located at the upper portion of the screen layout and may be docked or undocked based on the user's preference. It contains nine standard pull-down menus that provide access to the majority of the MicroStation dialogs, tools, and settings. Additional software products may add additional menu options such as the *Applications* pull-down for GEOPAK. The top portion contains information about the active file, including file path and name of the active design file, active model name, type of model, active design file format and the version of MicroStation loaded.

By default the Primary Tools and Attributes toolboxes display docked in this upper portion when a new MicroStation session is opened. The user may modify what toolboxes are displayed in this upper portion by selecting on the MicroStation menu option: **Tools**, selecting any toolbox and then docking that toolbox to the upper portion. Each toolbox can be customized as well by right clicking on the toolbox and selecting or deselecting in the tools popup window.

TASKS DIALOG

MicroStation supports task-based workflows, predefined or customized. A task is a set of any tool references grouped to facilitate a particular job, which can be grouped into standard toolboxes, custom toolboxes, or a combination of both types to suite any production need. The Task dialog, usually located docked to the left of the MicroStation window, displays those tools, tasks, and workflows for easy selection. A detailed discussion is found later under the Accessing Tools Section of this chapter.
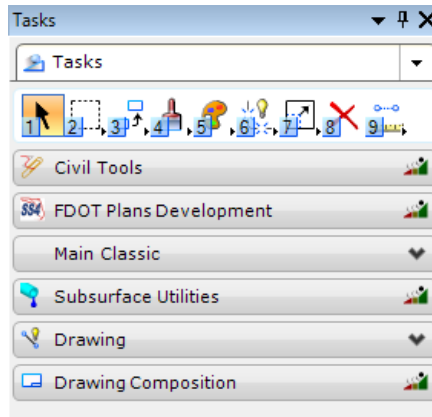
VIEW/APPLICATION WINDOW

The View Window is the area where the design elements are displayed. MicroStation allows a maximum of eight view windows to be open at any one time. The arrangement of the view windows can be customized within the application window.
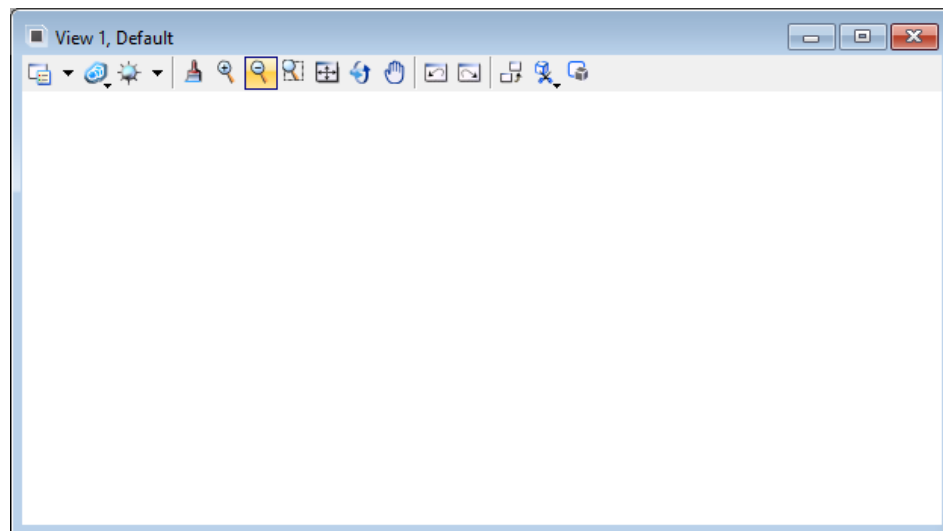
VIEW CONTROLS

Each view window has its own view controls to View Attributes, Update View, Zoom In/Out, Window Area, Fit, Rotate, Pan, View Previous and Next, Copy, Display Mode, Clip Volume and Mask. By default these are located at the top left of the MicroStation view window.

Note   View controls are discussed in detail in Chapter 3.

      FDOT MicroStation Essentials - Part I

VIEW GROUP TOOLBOX

A View Group is a named collection of eight view windows which allows for the set up of the desktop to display preferences including number of open view windows, window size and view orientation. Each view group is associated with a model making it easy to access and navigate through different models in the active DGN file, via the View Group drop-down menu. By default this is docked at the bottom of the Layout Dialog above the Status Bar.
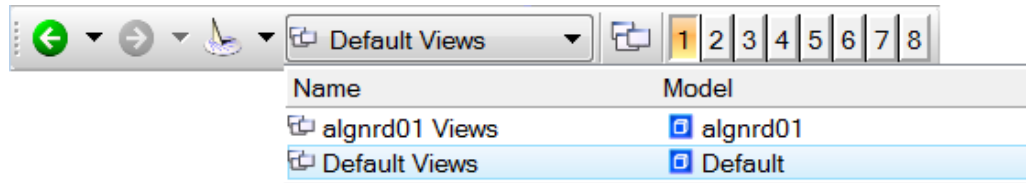
ACCUDRAW TOOL SETTINGS

This is used to facilitate data point entry. The AccuDraw Tool Settings opens or closes when clicking the *AccuDraw* icon on the *Primary Tools* toolbox. If using the default Function key menu, pressing *<F11>* opens the AccuDraw Tool Settings. If it is already docked or open, pressing <F11> sends focus to the AccuDraw Tool Settings.
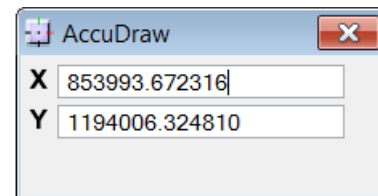
AccuDraw (Docked)   *AccuDraw (Floating)*

STATUS BAR

The Status Bar is located across the bottom of the screen layout and provides prompts, context-sensitive assistance and some interactive functionality, which are described next. The label for the Status will display by hovering the cursor over any box. The user may customize the Status Bar by *right-clicking* on it to open a check list of options to check on or off for display.

Element Selection > Identify element to add to set   Q
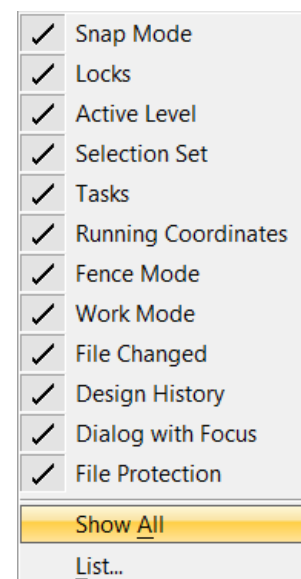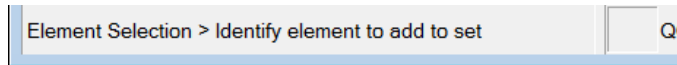
Selected Tool > Prompt shows the name of the selected tool or view control and (usually) a prompt for the next step in the normal procedure for using it.

Message Center shows the system message information. Clicking in the Message Center portion of the status bar *(field following the >Prompt)* opens the Message Center window. This window contains a running log of system messages and any further description about the message if applicable.

Message Center

Line String, Level: Building_ep, Ref: survey topo (TOPORD01.DGN)
Line String, Level: 60, Ref: survey topo (TOPORD01.DGN)

Message Details
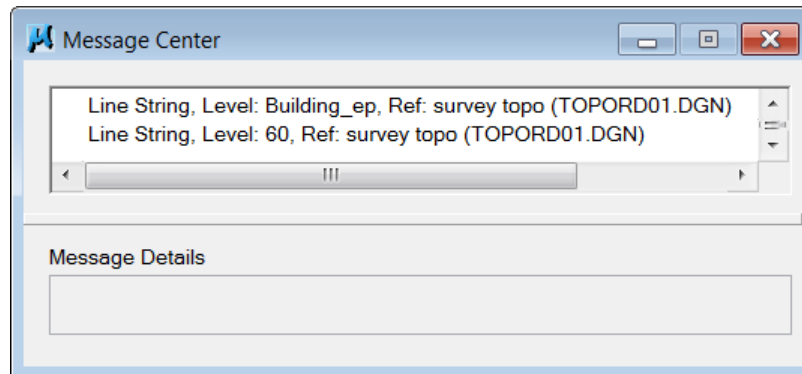
Snap Mode shows the type of snap selected with the associated icon. Clicking the Snap Mode field posts the pop-up Snap Modes menu. The effect is the same as pressing the Tentative button while holding down the <Shift> key.

Button Bar
AccuSnap
Multi-snaps

Nearest
● Keypoint
Midpoint
Center
Origin
Bisector
Intersection
Tangent
Tangent Point
Perp
Perp Point
Parallel
Point Through
Point On
Multi-snap1
Multi-snap2
Multi-snap3
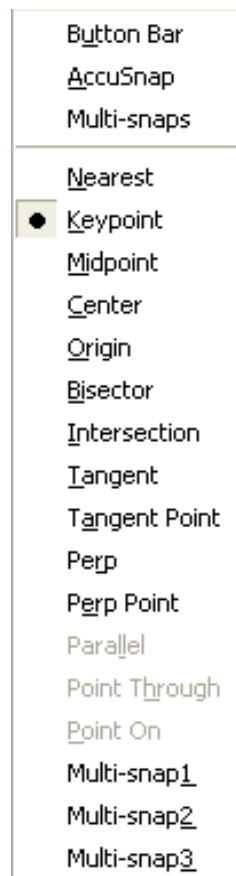
Active Locks icon provides access to the Locks submenus. (It does not indicate status information.)

Active Level area indicates the Active Level setting. Clicking the Active Level field opens the Level Manager dialog, which is used to control level display and level symbology for the open DGN file and attached references.

Selection Set, if shown, indicates elements are selected and displays the number of selected elements. If no elements are selected, this field on the Status Bar is blank. Clicking the status bar when it shows the number of selected elements opens a pop-up menu containing the following items:

Element Selection pop-up menu

Pop-up menu item	Choose to
Select Element	Select an elements in the active model.
Select All	Select all elements in the active model (effect is same as choosing Edit > **Select All** or pressing <Ctrl-A>).
Select None	Deselect all elements in the active file (effect is same as choosing Edit > **Select None**).
Select By Attributes	Open the **Select By Attributes dialog** (effect is same as choosing Edit > **Select By Attributes**).

Tasks (Hidden by default) shows the name of the current task.

Running Coordinates displays the longitude and latitude coordinates of the current position as the cursor moves across the view. Click the box to open the pop-up menu of options

Pop-up menu item	Description
Position	Displays the X, Y, and Z coordinates of your current position.
Delta	Shows the X, Y, and Z displacement from the last data point.
View Delta	Shows the view coordinate system displacement from the last data point.
Distance	Shows the distance and direction from the last data point.
ACS Position	When a tentative point is entered, the readout also shows longitude, latitude.
ACS Delta	Shows the auxiliary coordinate system displacement from the last data point.

Fence Indicator, if shown, indicates the fence mode of the fence that was placed. If there is no fence placed in the design file, this field on the Status Bar is blank.

DGN Work Mode indicates which Work Mode is in effect. MicroStation natively reads and writes both the MicroStation DGN file format and the DWG file format thus enabling less data loss from sharing. DGN Work Mode is indicated with a blank field.

- **DWG Work Mode** is a working environment within MicroStation specifically adapted to the editing and reviewing of DWG files. In DWG ("DWG") mode, certain functionality is disabled by default in order to restrict MicroStation to creating only information that can be stored in DWG format. A DWG file can be directly referenced from a MicroStation DGN file.

File Changed, if shown in the lower right-hand corner, indicates that the DGN file has been modified during the current session. If the preference *Automatically Save Design Changes* is off, it indicates that there are changes that have not been saved. If the diskette is red and has an "X" through it, it means that the file is read-only.

Design History is indicated by an icon in the status bar. If design history is not initialized, the icon is dimmed. If design history is initialized but there are uncommitted changes, a pencil is superimposed on the scroll. Clicking this icon opens a pop-up menu containing the following items:

	Initialize
	Delete
	Check for Changes
	Conflicts
	Commit
	Design History
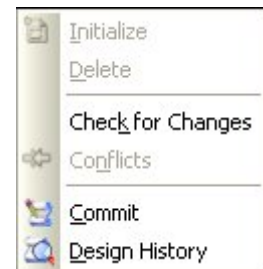
Input Focus indicates the focus location. MicroStation follows actions and attempts to move input focus for common operations; for example, when placing a line the focus moves from Home to the AccuDraw dialog, then back to Home.

Focus Location	Indicates	Displays on status bar
home	you are at the top level	
Tool Settings	Tool Settings window has the focus	
Key-in	Key-in window has the focus	
AccuDraw	AccuDraw window has the focus	
(No icon)	another window or dialog has the focus	

Digital Rights indicates if the file is protected and digitally signed. If none exists, then the box will be blank.

Note   Caution must be advised in using this option.

- When a pre-File Protection version of MicroStation V8 tries to open a protected file, it recognizes the file as a V8 encrypted file. MicroStation displays an explanatory message but will not open the file.

- The MicroStation status bar displays a signature icon when a file contains any file or model signatures.

STANDARD FDOT MENU

The FDOT Menu is part of FDOT's delivered customized workspace. It is a floating menu bar that contains a collection of standard and discipline specific menus and command buttons designed to assist in plans production for FDOT projects. The FDOT Menu automatically loads when the File Open dialog *Interface* is set to *FDOTSS4* and is used to activate commands inside MicroStation through an efficient, portable, pull down menu interface that accesses all FDOT CADD standards and customization.

The Standard FDOT Menu is the base menu setup from which all selected discipline specific menus build upon. The menu bar settings, such as display color, opacity or menu location, can be modified by the user by right clicking anywhere on the menu bar to open a popup menu.

Note   The FDOT (right click) Menu option: Save Settings must be selected to save any display modifications for this menu for future MicroStation sessions. The MicroStation menu option: File > Save Settings does not apply to the FDOT Menu settings.
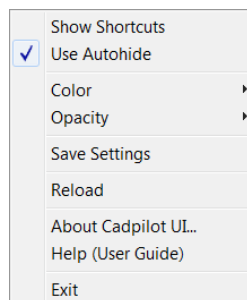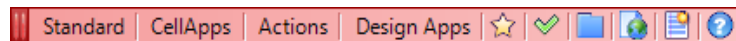
Note   The Help icons located by any FDOT Menu option provide links to associated Help files.

STANDARD MENU

Provides links to the FDOT software versioning, Discipline configuration access, ECSO Training/Support links, external resource links and Level Library attachments. *Help* icons located by any FDOT Menu option provide links to associated *Help* files.
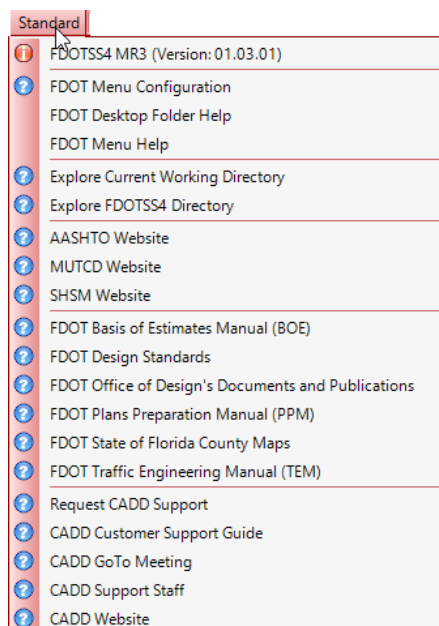
CELLAPPS MENU

Provides **Cell Utility** tools for easy access to select and place cells from the many Cell Libraries delivered and supported by FDOT. *Help* icons located by any FDOT Menu option provide links to associated *Help* files.
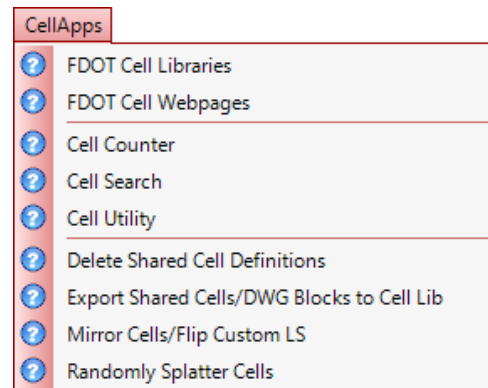
ACTIONS MENU

Provides links to FDOT Tools and Utilities commonly used by all disciplines, such as Project/File creation, Adhoc Management, and Sheet Clipping & Labeling. *Help* icons located by any FDOT Menu option provide links to associated *Help* files.
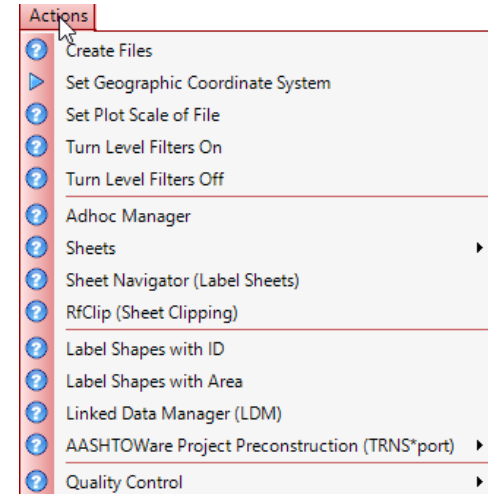
DESIGN APPS MENU

Provides links to additional FDOT design-related apps commonly used by all disciplines, such as File Cleanup, Custom Line Style Apps, Reference File Apps and more. *Help* icons located by any FDOT Menu option provide links to associated *Help* files.
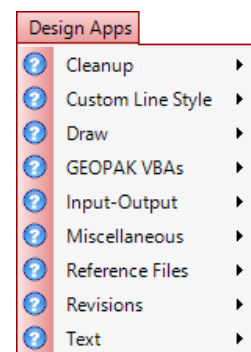
DISCIPLINE MENU

The standard FDOT Menu is the basis on which the user can choose to build configurations for FDOT Discipline specific menu options. One or more may be selected through the FDOT Menu option: **Standard > Configuration** to display and provide links to FDOT tools and utilities directly associated to those specific disciplines for the ease of selection by the user. Note that icons located by any FDOT Menu option provide links to associated Help files.
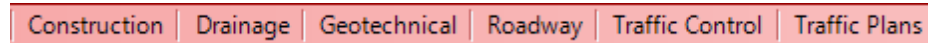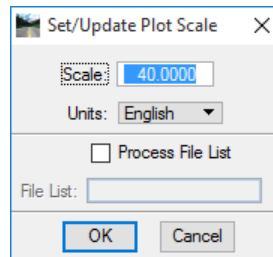
| Construction | Drainage | Geotechnical | Roadway | Traffic Control | Traffic Plans |

COMMAND BUTTON OPTIONS

Run Plotscale opens the Set/Update Plot Scale dialog to establish Plot Scale.

Set/Update Plot Scale ✕

Scale: 40.0000
Units: English ▼
☐ Process File List
File List:
OK   Cancel

Run QC Quick opens the QC Software for interactive reporting and correction of level and symbology for the active design file in compliance with the FDOT CADD Standards.

QC Quick (v2015.1.6.0) — ☐ ✕

QC Compliance: 99% (3 errors out of 2505 elements checked) Rule: dsgnrd   QC Year: current

Critical File Status: Critical

	Model	Element Type (Level)	Invalid Level	Invalid Color	Invalid Weight	Invalid Linestyle	Feature Definition
▶	Default	ComplexShap...	ConcSlabs_ep				
	Default	Ellipse (Constu...	Constuction				
	Default	Text (Constuct...	Constuction				

☑ When correcting levels, set all symbology to ByLevel

Level   : ConcSlabs_ep

Color   : ByLevel

Line Weight : ByLevel

Line Style : ByLevel

Hilite Color

Close

Show Command List opens the Cadpilot UI TabList table which provides listings of discipline related Level Names and Level Descriptions for selection.

| Level Name	Description
AdhocPoint_dp	Adhoc Points Placed by SheetInfo
ATDims	AutoTurn Profile Dimensions, Tire Envelope
ATText	AutoTurn Profile and TT Dimensions, Annotation, TT Overhang
ATVHP	AutoTurn Temporary Profile for Building Custom Type
BLSurvey_dp	Baseline Survey
BreakLine_dp	Break Lines
CLConst_dp	Center Line of Construction
ClearSight_dp	Clear Sight Lines and Limits
ClearZone_dp	Clear Zone Elements
ClipBorder_dp	CLIP Border
Cloud_dp	Construction Cloud
COGO_dp	COGO Information
ConstArea_dp	Construction Area Crosshatch
ConstLimits	Construction Limits
ConstLines	Construction Lines and References
ControlZone_dp	Control Zone
CurveData_dp	Curve and Coordinate Data Elements including PC, PT, PI sym...
CurveDataLabel_dp	Curve Data Labels PC,PI, PT
DimLines_dp	Dimension Lines, Corresponding Terminators, Equation Lines
FLMap1	Florida Map items - weight of 1
FLMap2	Florida Map items - weight of 2
FLMap3	Florida Map items - weight of 3
GridMaj_dp	Grid Lines Major in Profile and Cross Section

Tabs: SignalsSymbology, TrafficControlSymbology, Drafting-Symb, Drainage-Symb, SigningPMSymbology, Roadway-Symb, Lighting-Symb

Open Browser opens the Internet Browser window. This Command button remembers the last browser window previously opened and returns to that window when executed.

Cadpilot UI - Internet Explorer

File

ABOUT:BLANK

Done
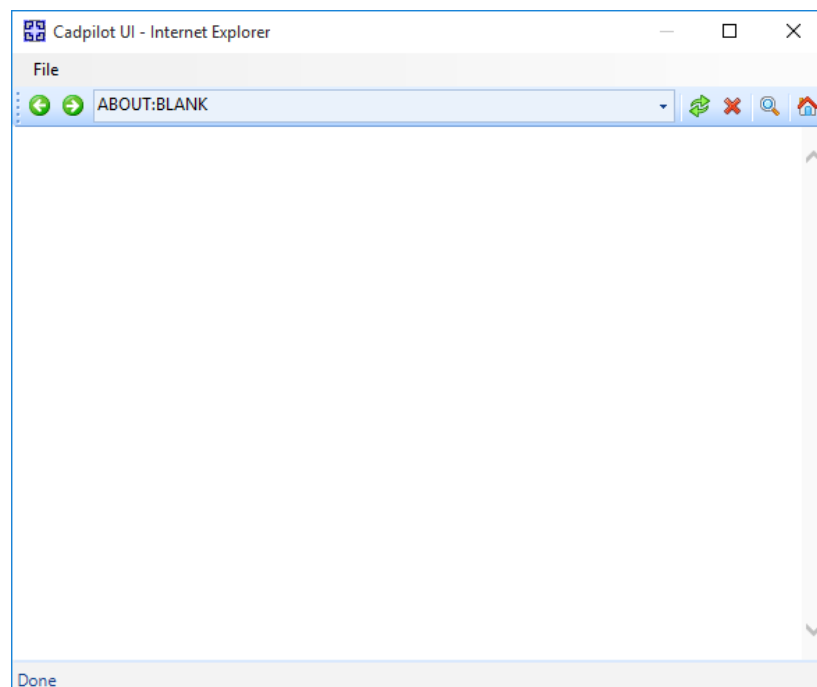
Label Sheet (Sheet Navigator) opens the Sheet Navigator dialog. Sheet Navigator can be also accessed from the FDOT Menu option: **Actions > Label Sheet (Sheet Navigator).**
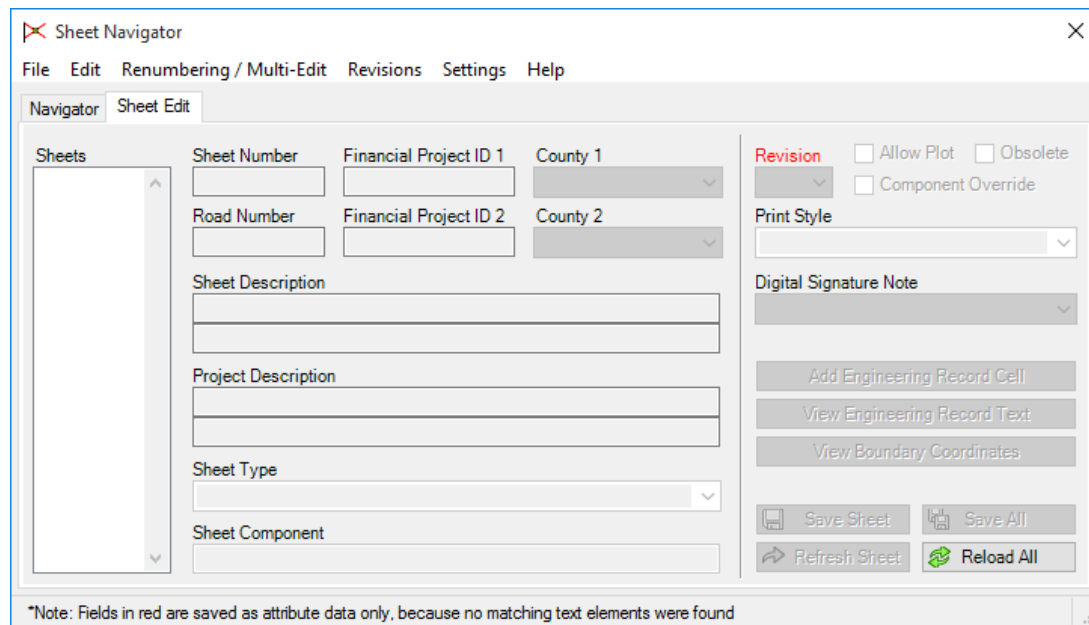

MICROSTATION DRAWING TOOLS

PARENT-CHILD RELATIONSHIP

The MicroStation toolboxes are hierarchical; that is, there are "parent-child" relationships between toolboxes. The "parent" toolbox displays one tool from each of its "child" toolboxes. The toolboxes can be closed by clicking on the close button ( ) on the far right side of any toolbox or by selecting the MicroStation menu option **Tools > Close Toolboxes**. Any tool can be selected from a "child" toolbox thorough the dropdown menu of the "parent" toolbox without floating the "child" toolbox. This aspect of toolboxes helps minimize the screen space.
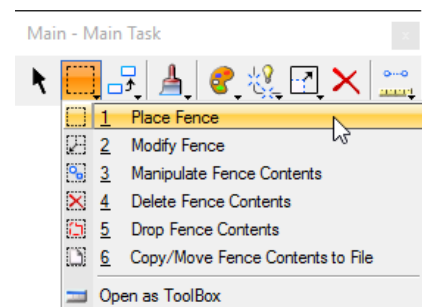

DEFAULT TOOLBOXES AND DIALOGS

When starting MicroStation for the first time, the Attributes toolbox, Primary toolbox, Main task and Tasks dialog are opened by default and docked. The Main task is embedded at the top of the Tasks dialog. The Attributes and Primary toolboxes are docked at the top of the MicroStation window along with the MicroStation Menu. These toolboxes control many of the settings for creating elements and individual tools. If not displayed when MicroStation is first launched, the user can open them by selecting **Tools** from the MicroStation menu.
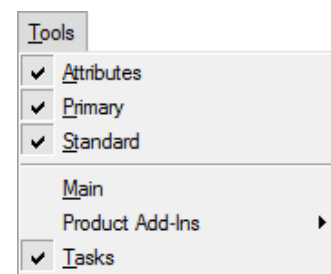
ATTRIBUTES TOOLBOX

The Attributes toolbox contains controls for setting the active element attributes that control the appearance of elements in a design file. These attributes include color, line style and weight, and are collectively referred to as symbology.
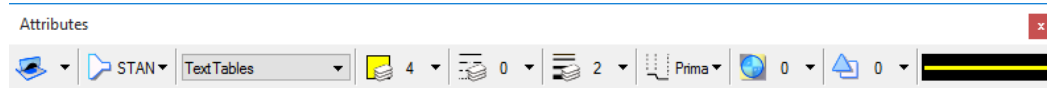
While symbology can be set per element, FDOT uses the **ByLevel** feature in MicroStation and has created a set of CADD Symbology Standards. FDOT has a standard listing of Levels preset with ByLevel Color, Line Style and Weight symbology. Users need only change the active level to change the active symbology. These Levels are grouped into specific FDOT Standard Rules which are used for compliancy checking of each FDOT standard design filename.

To override the **ByLevel** setting of one of the symbology components, users can select an option from the Attributes Toolbox and choose the new symbology from the drop-down list.

Note FDOT does not recommend overriding the ByLevel symbology settings. This will cause catastrophic consequences when other tools using symbology perform other functions. This is especially true when using the GEOPAK application used by FDOT in conjunction with MicroStation for surveying and roadway design.

All tools in a toolbox are not always visible by default. A simple right-click on the toolbox will display a listing of available tools to populate the toolbox.

PRIMARY TOOLS TOOLBOX

The Primary Tools toolbox provides easy access to frequently used MicroStation tools that are production oriented such as Models, References, Levels, and Cells. By default it is docked to the top of the MicroStation window (directly beside the menu bar).
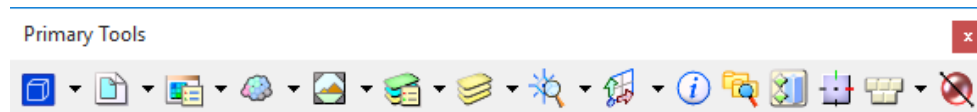
- *PopSet* - A tool that toggles on/off and is used to automatically prevent the display of a *Tool settings* dialog while not adjusting its controls. This is a great way to reclaim valuable screen "real estate" and reduce pointer movement. PopSet is enabled by default.
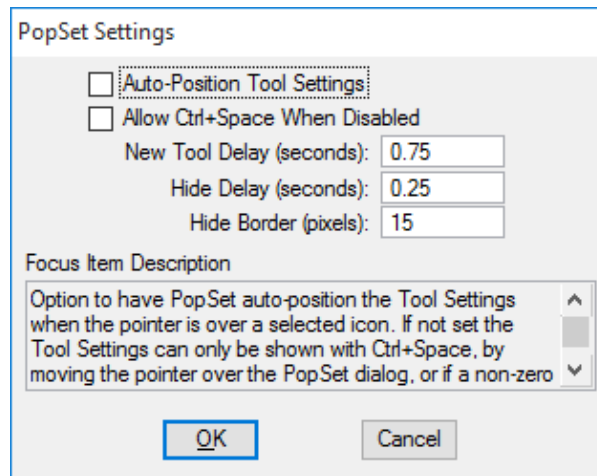

PopSet (Enabled)   PopSet (Disable)

Set *PopSet* settings by right-clicking on the PopSet Icon and selecting Properties.

- *__Key-in__* - A dialog that allows users to type in any MicroStation command, as well as browse through a command's options. While nearly all of MicroStation's commands can be accessed from either a toolbox or a pull-down menu, there are times when the user may want to type in a command.

    The **Key-in** icon contains an adjacent down arrow. Clicking on the down arrow opens the building dialog in a pop-up mode to build the key-in when unsure of the run command key-in information. It closes automatically when the pointer moves off the dialog.
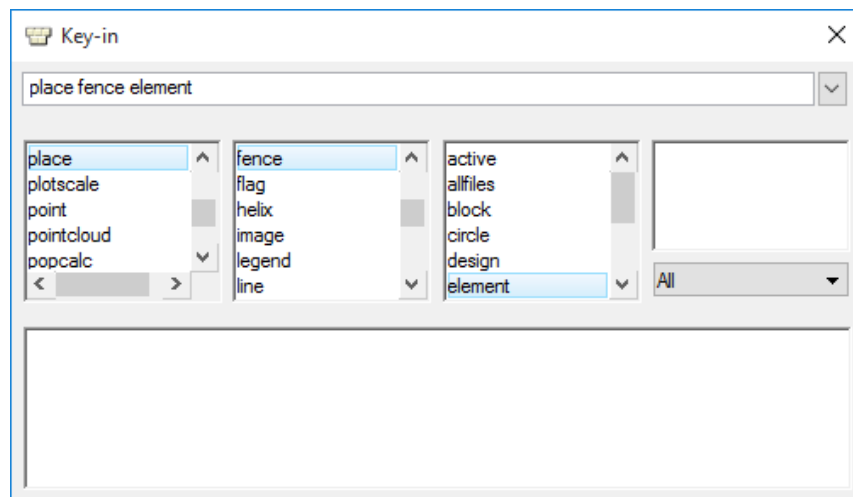
    Clicking on the **Key-in** icon opens a floating Key-in dialog with only the key-in field, which can be docked as shown below. The **Browse Key-in** icon ( ) found on this dialog is used to open the building dialog shown above.

If the Key-in dialog is open, additional settings are available to make the display options more user friendly. Right-clicking anywhere in the Key-in window, except the key-in field at the top, opens the Key-in Settings dialog.
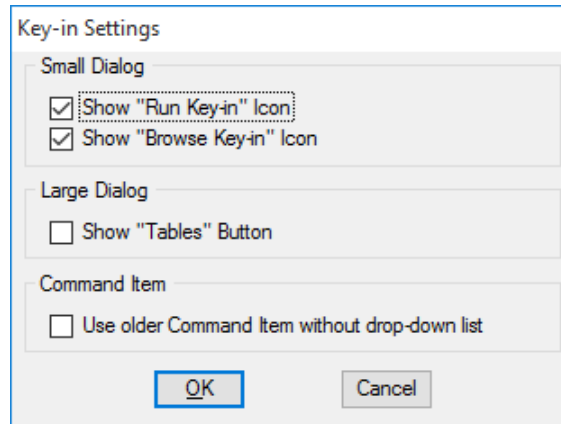
STANDARD TOOLBOX

The Standard tool bar provides standard Windows options: *New, Open, Save, Print, Cut, Copy, Paste, Undo, Redo, and Help*. This toolbox is not open by default.

MAIN TOOLBOX

The Main toolbox provides frequently used MicroStation tools used for element selection, manipulation, and modification. By default it is embedded as the Main task at the top of the Tasks dialog on the left of the MicroStation window (directly beside the view window). The toolbox can be opened by right-clicking on this task item and selecting *Open 'Main' as Toolbox* <OR> by through the MicroStation menu option *Tools > Main*.
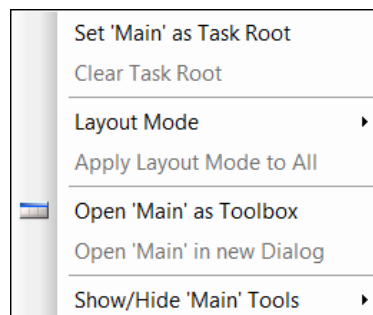
MicroStation offers *Positional Keyboard Navigation* (*Keyboard Input*). Each "Parent" tool option of the Main task is associated with a keyboard number (1 – 9) as with each of the "child" options. The user needs only to set the focus on the task bar and type the "Parent" number then the "child" number to access the specific tool. (See the section: *Accessing Tools* & Controlling *The Focus* below to see a chart of the Keyboard.)

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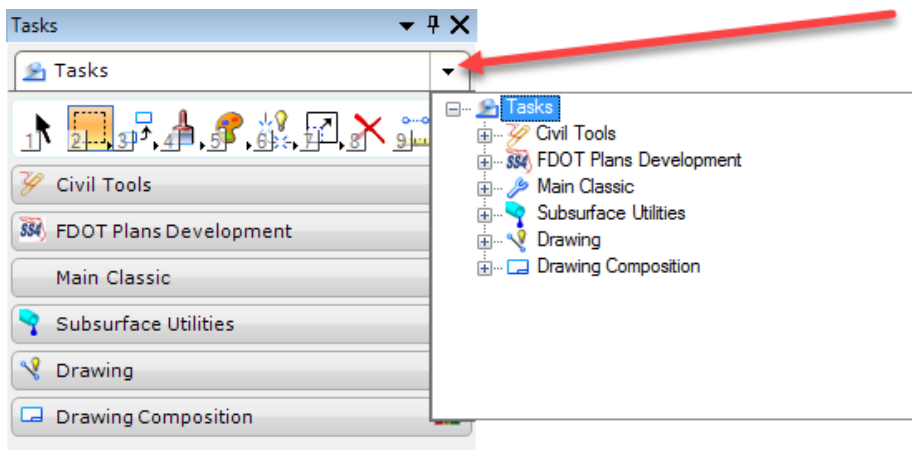
TASKS DIALOG

The Tasks dialog is docked by default on the left side of the MicroStation application window, but can be docked to the right by selecting the title bar and dragging the cursor over a docking indicator [image] . The Tasks dialog may also be hidden (minimized) by selecting the Auto-Hide (push-pin) [image] icon. Hovering the cursor over the minimized Tasks opens the Tasks window for selection. Reselecting the Auto-Hide (push-pin) [image] icon restores Tasks to an open display.
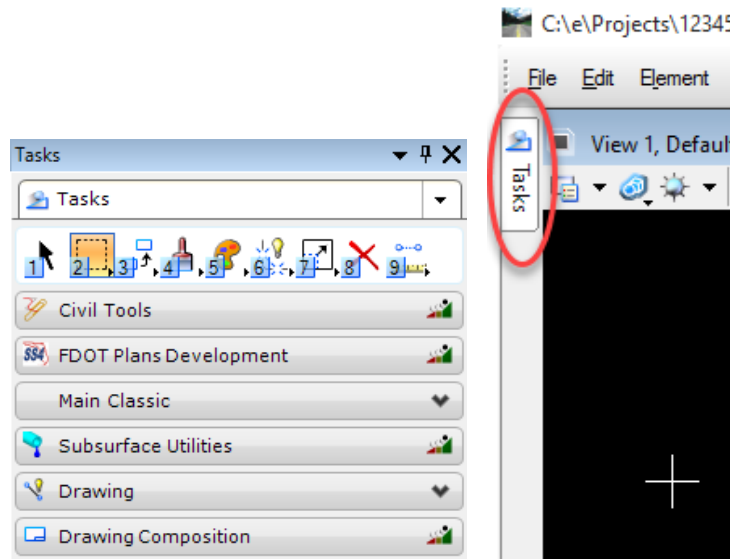
The previous drill down version of the Tasks Navigation tool is still available for navigational search by selecting the arrow to the right of the *Task* tab.

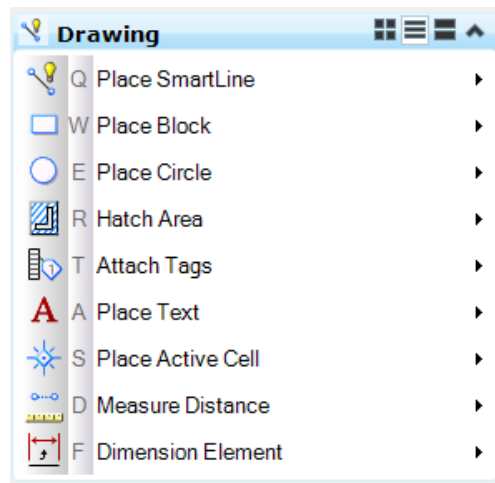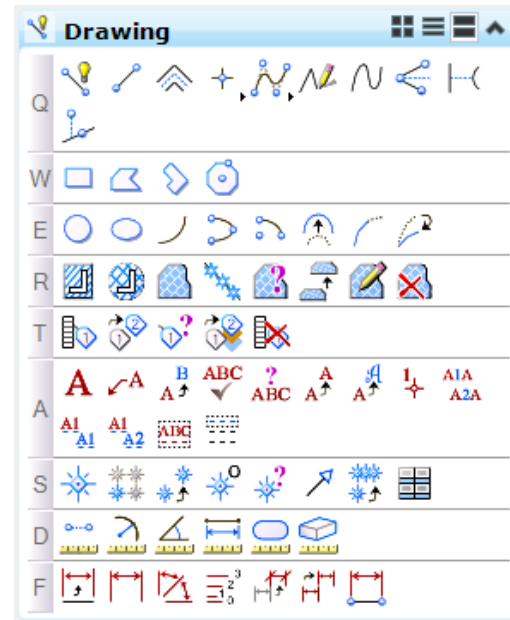Each task item listed in the Tasks dialog extends to display the tasks groupings in three Layout Modes: Icon, List and Panel by way of selecting the applicable   icon located on each Task title bar.
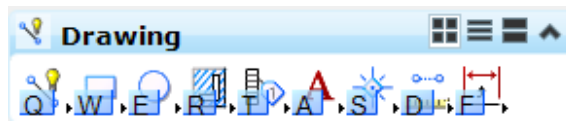

Icon Layout

List Layout   *Panel Layout*

MicroStation provides Positional Keyboard Navigation with mapping each Task "Parent" entry to a *letter* on the keyboard. Each "Child" tool within the toolboxes is mapped to a *number or letter*. With the Focus placed on the Task, the user may type the "Parent" letter then the "Child" number to execute the tool. For example: after placing the focus by selecting the Drawing Task group, the user my type '*Q*' then '*1*' to execute the *Place Smart Line* tool.

Each Task may be opened as a toolbox by *right-clicking* on the Task and selecting *Open…. as Toolbox* from the pop-up menu. As with the Icon Layout, these toolboxes have arrows to open dropdown selection lists of tools.

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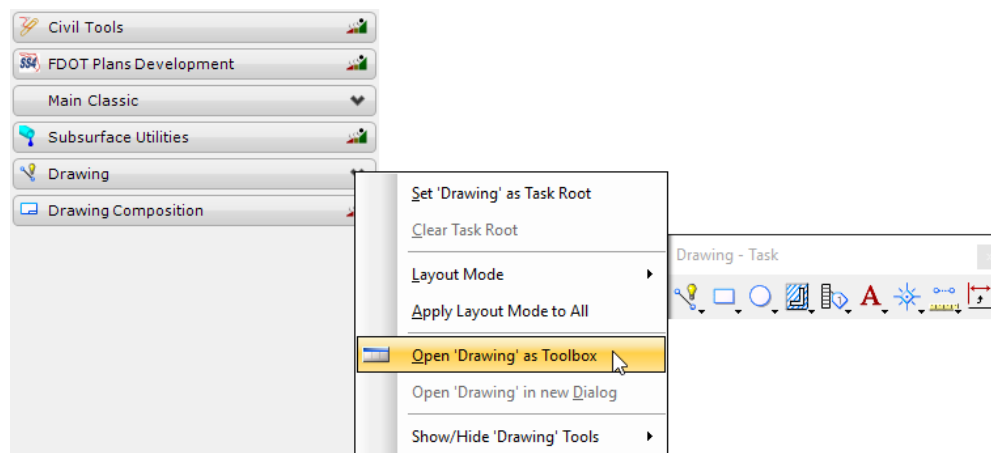
The FDOT Workspace provides a customized FDOT *Task Workflow* setup to assist the user in meeting the FDOT CADD Standards for specific disciplines. Upon selection of the FDOT workflow, an additional *FDOT Task Tab* displays at the top of the Tasks dialog and opens for selection. Note that each of the FDOT Tasks may be opened as floating toolboxes as well.
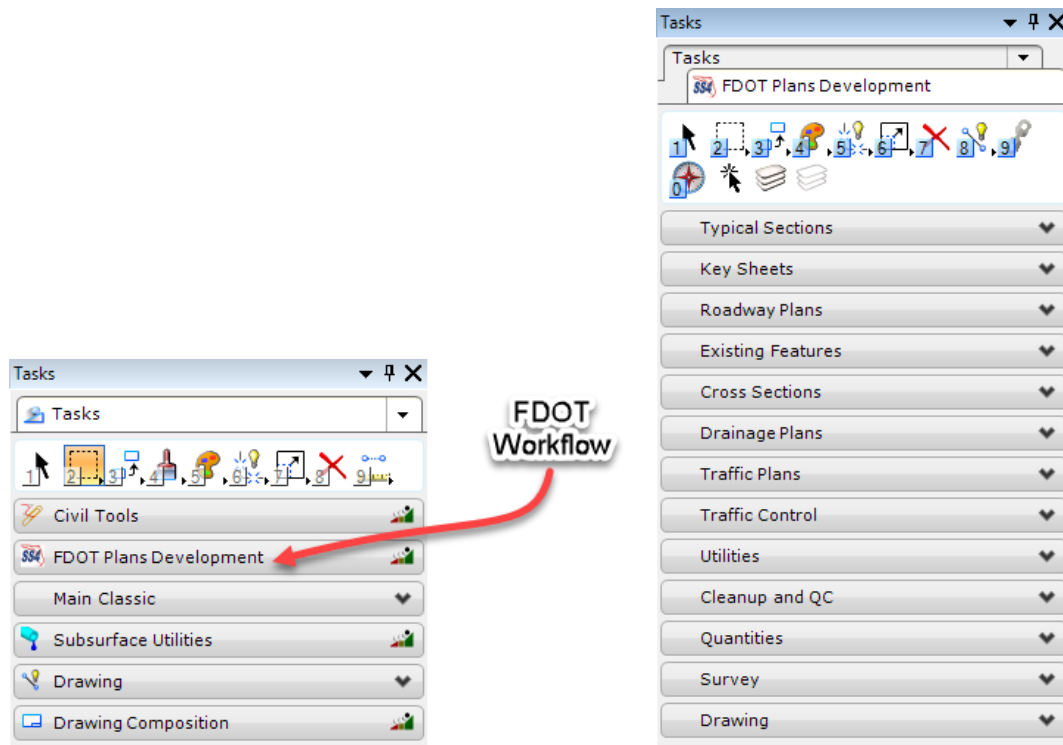
DISPLAY / HIDE TOOLS

MicroStation has dozens of drawing tools. These drawing tools are grouped and displayed for convenient selection in *Toolboxes*. The toolboxes can be grouped and referenced to facilitate a particular job to form a collection called a *Task*. A sequence of tasks can be grouped into *Workflows*. All toolboxes and tasks can be customized. MicroStation provides many preset toolboxes for the users. FDOT has also customized many toolboxes.

Note   Only one task is active in MicroStation at any given time.

Tools are represented in toolboxes by icons. For simplicity, the term "tool" is used to refer both to a tool and its icon. The arrangement of tools in a floating toolbox can be changed by resizing its window.

MicroStation toolboxes and any specific tool can be displayed or hidden by:

- Opening the Tool Boxes dialog to select or deselect toolboxes to display or hide by selecting from the MicroStation menu option *Tools > Toolboxes* at the bottom of the drop down list.
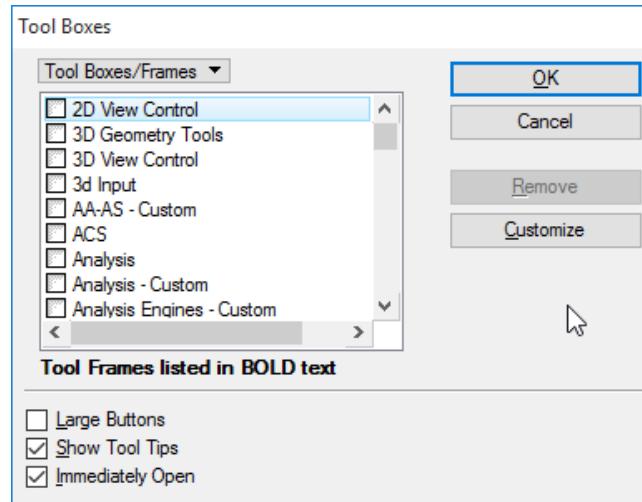
- Opening the Show/Hide Tools dialog to select or deselect desired tools of a specific toolbox to show or hide by right clicking on any Toolbox and selecting the last option on the fly out menu, List.
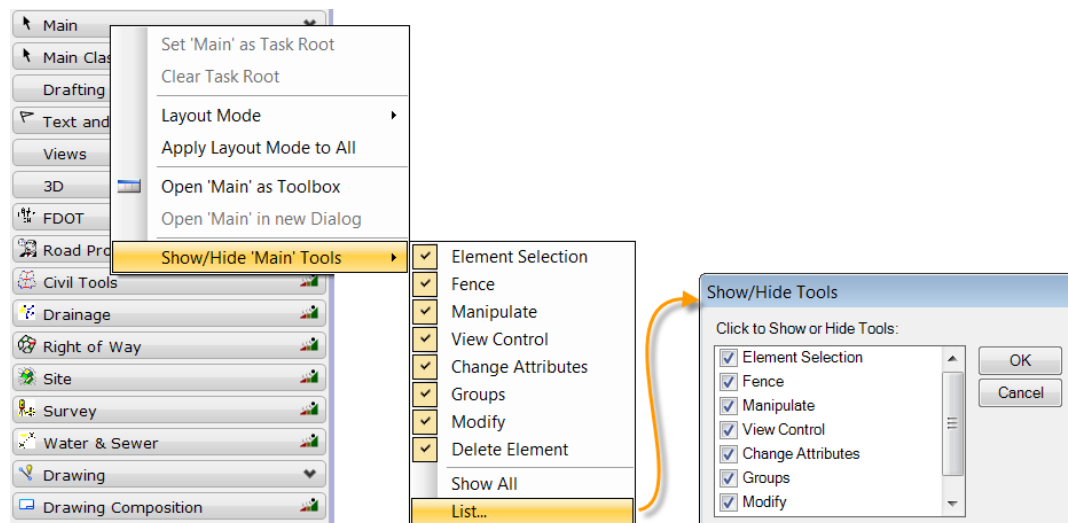
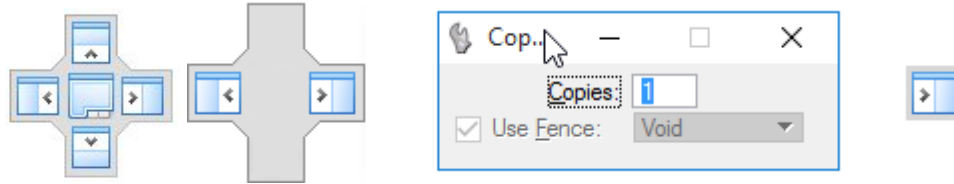### DOCKABLE DIALOGS

MicroStation allows for many dialogs to dock on the edges of the application window to save screen space. When a dockable dialog is dragged across the screen over any docking indicator, a transparent box displays towards the side allowable for docking. The user needs only to drag a dockable dialog over one of the indicators where the cursor touches the desired indicator, and then release it to attach to selected side.


The following is the list of dockable dialogs, and which edges they can be docked to:

- *Tool settings* window (left or right)

- Element Information dialog (left or right)

- Project Explorer dialog (left or right)

- Link Sets dialog (left or right)

- Tasks dialog (left or right)

- Markups dialog (left, right, top, or bottom)

- Item browser (left or right)

- Details dialog (left, right, top, or bottom)

- Feature Manager dialog (left or right)

- References dialog (top or bottom)

- Raster Manager dialog (top or bottom)

- Models dialog (top or bottom)

- Saved Views dialog (top or bottom)

- Level Manager dialog (top or bottom)

- Level Display dialog (left, right, top, or bottom)

Note	These dialogs can be found on the Primary Tools toolbox. The way in which one of these dialogs is docked differs from the way in which a toolbox is docked.

Once a dockable dialog is docked, the user can hide it by clicking the Auto Hide (push pin) icon and minimizing it to a tab. The user needs only to hover the cursor over the tab to open the dialog for selection.
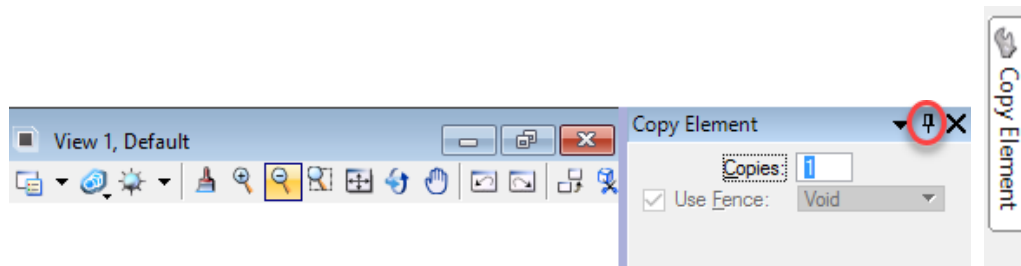
DOCKABLE TOOLBOXES

MicroStation allows toolboxes to be docked along any edge of the application window. The user needs only to drag the toolbox by the title bar toward the edge of the window to allow it to dynamically jump to an appropriate sized docking area. When docked, the title bar does not display and borders become narrower. A single dragging operation can be used to undock by placing the cursor on the toolbox's narrow border or on a blank area and dragging the box away from the edge.

Note   To move a toolbox or other dockable window to a docking area without docking it, hold down the <Ctrl> key while dragging the title bar.

KEYBOARD INPUT & CONTROLLING THE FOCUS

In Microsoft Windows, the *Input Focus* (keyboard focus) is defined as the location to which the keyboard input is directed. In MicroStation there is a hierarchical focus model with *Home* as the top level. The current focus location is indicated in a *Status Bar* field at the bottom of the screen layout. The user needs only to hover the cursor over this field to determine location of the focus.

When the focus is *Home*, the user does not need to use the mouse to navigate toolboxes, tasks or the Tools Settings window. MicroStation provides *positional keyboard navigation* for the user with easy keystrokes using *letters* and *numbers* to select tools and adjust *tool settings* controlled through option menus and check boxes.

Note   Selecting the <Esc> key on the keyboard brings the focus back to Home. It also reveals mapping codes on tool icons and tool setting windows.

MicroStation position maps the keyboard by default with the following position mapping:

Default position mapping.

The **X** keys (<1>, <2>, <3>, <4>, <5>, <6>, <7>, <8>, <9>, and <0>) are mapped to the icons in the Main task.

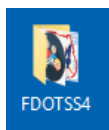The **O** keys (<Q>, <W>, <E>, <R>, <T>, <A>, <S>, <D>, <F>, <G>, <Z>, <X>, <C>, <V>, and <B>) are mapped to the icons in the Tasks dialog.
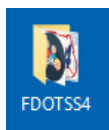
The **** keys (<Y>, <U>, <I>, <O>, <P>, <H>, <J>, <K>, <L>, <;>, <N>, <M>, <,>, <.>, and </>) are mapped to the controls in the Tool Settings window.

Exercise 2.1 *Reviewing the Design Environment*

➢ ***Open a*** MicroStation ***Session in FDOT Workspace***

1. Click on the FDOT folder ![FDOTSS4] to open the FDOT Desktop window and review options.

   a. Except for the FDOTSS4 icon, double-click the various other icons to open their respective windows and then select the **Exit** ![X] icon from the upper right corner of each window to close, when review is complete.
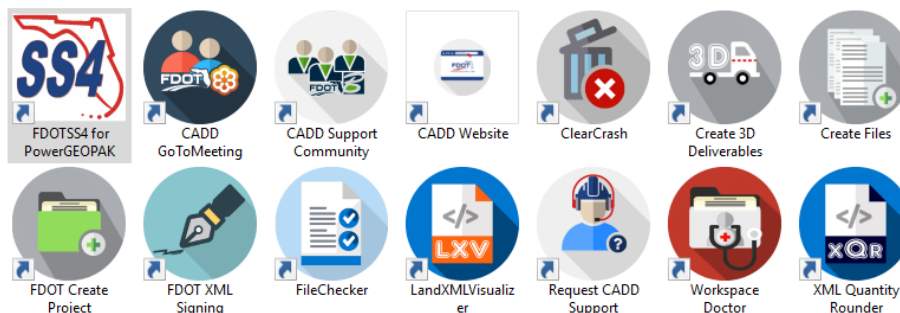

2. To open MicroStation in the FDOT Workspace, double-click on the **FDOTSS4** icon. The File Open dialog displays for review and setup.

   a. Set the FDOT workspace settings: *User* set to **FDOTSS4;** *Project* set to **12345678901;** and *Interface* set to **FDOTSS4.**

   b. Set *Files of type* to **CAD Files (*.dgn,*.dwg,*.dxt)**

   c. Select the design directory in the *Look in:* field by using the dropdown arrow next to the field to navigate to the e\projects\12345678901\roadway directory. Once selected the *Name* list box should populate.

   d. Select *File Name*: **dsgnrd01** from the list. A preview of selected file will display on the right.
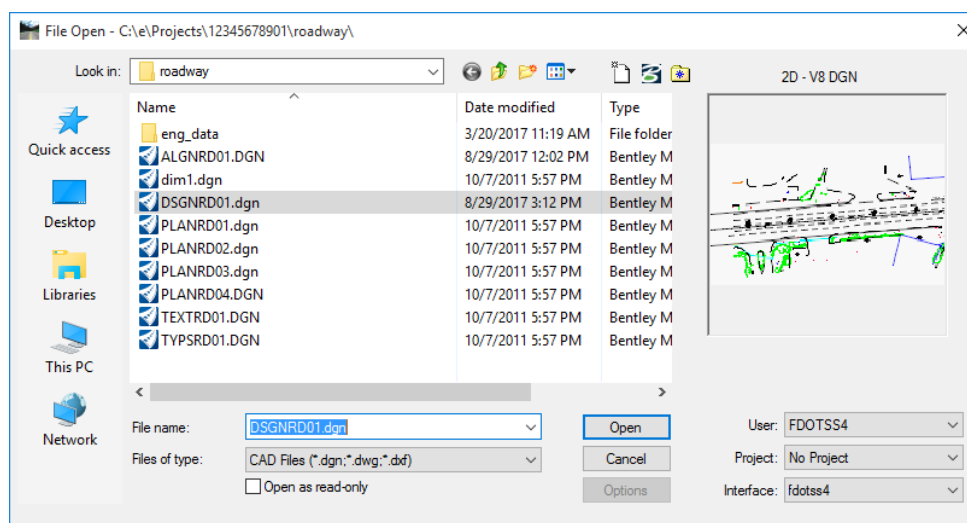

3. Click **Open**. A MicroStation session will open with selected file.

4. Review the MicroStation application window:

    a. Title Bar with all the file and setting information;

    b. Default toolboxes docked at the top with the standard MicroStation menu;

    c. Default Tasks dialog docked to the left side;

    d. Default Element Selection Tool settings window;

   e. Default View Groups and AccuDraw docked to at the bottom along with the Status Bar.
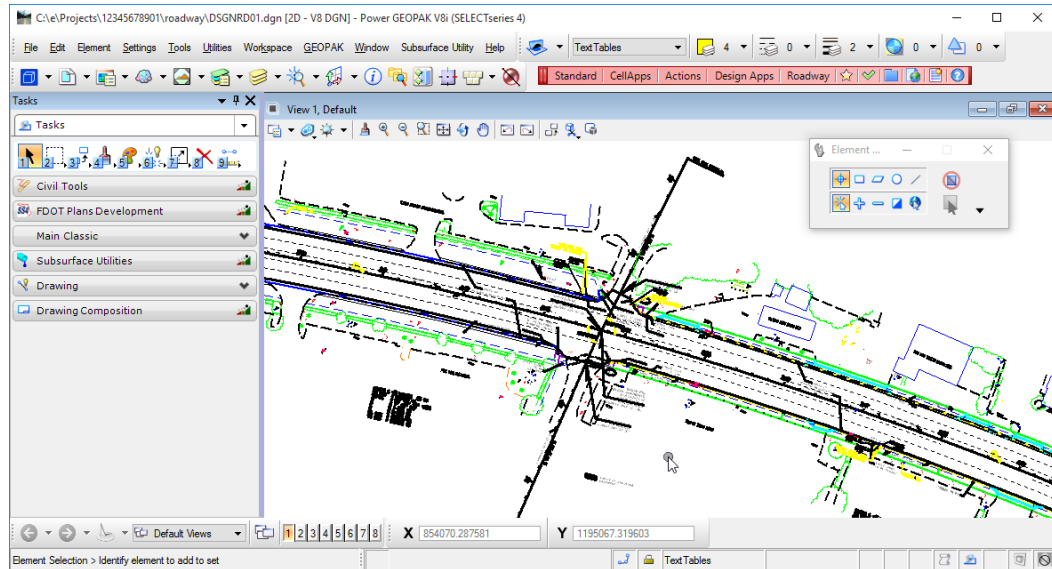
5. From the MicroStation menu, select **Settings > Design File**…. The Design File Settings dialog will display for review.

    a. Select each *Category* item to review the changing settings pane for each.

    b. Select **Working Units** *Category* and confirm/set the settings as shown below.
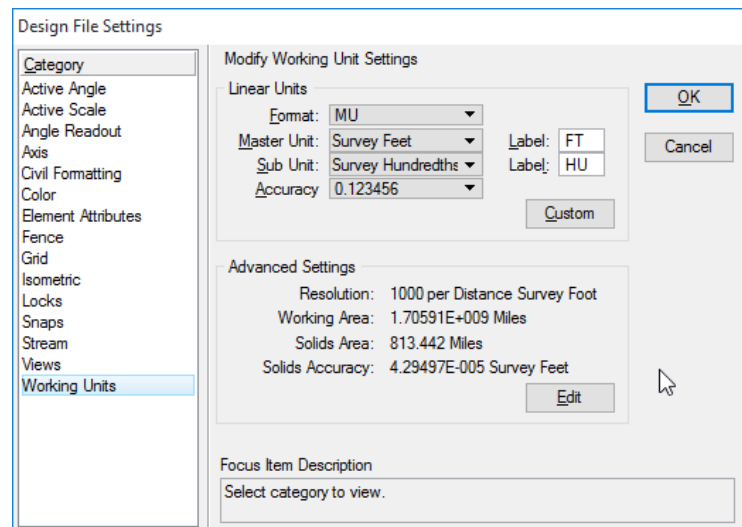
6. Click **OK** when completed.

7. From the MicroStation menu, select **Workspace > Preferences…** The Preferences dialog will display for review.

    a. Select each Category item to review the changing settings pane for each.

    b. Select **Operations** Category and confirm/set the settings as shown below.
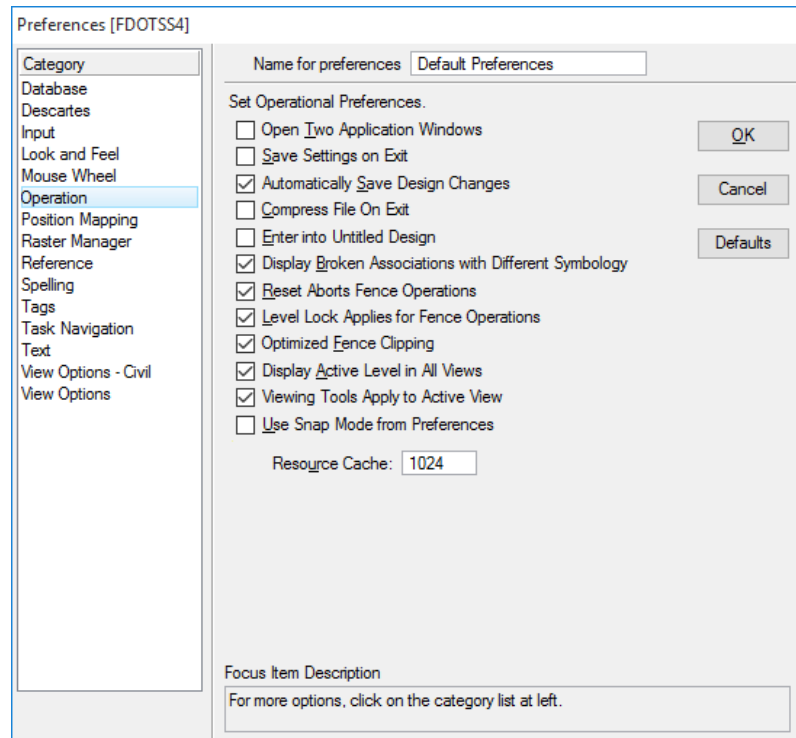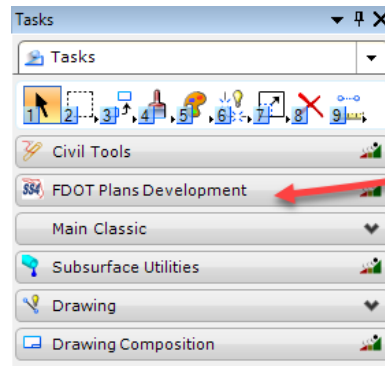

8. Click **OK** when completed.
9. Do not Exit MicroStation.

Exercise 2.2 *Accessing and Displaying Tools and Toolboxes*

➢ *Exploring the Tasks Dialog*

1. Continue with *dsgnrd01.dgn* for this exercise.

2. From the `Tasks` dialog, select the **FDOT** Plans Development Task Workflow. Notice that an additional FDOT Plans Development task tab displays with the preset FDOT tasks.

3. Select the down arrows of Roadway Plans on the FDOT tasks to review the tools.
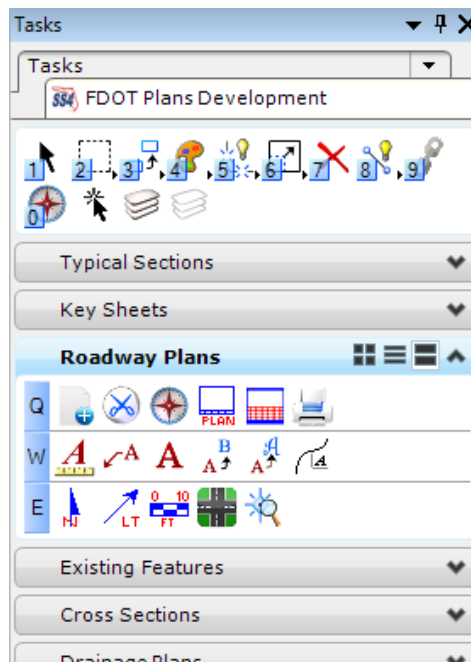
4. Click on the FDOT Tools task bar and select each of the Layout Modes icons.
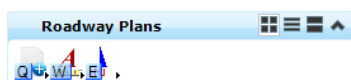
Icon Layout Mode	*List Layout Mode*	*Panel Layout Mode*
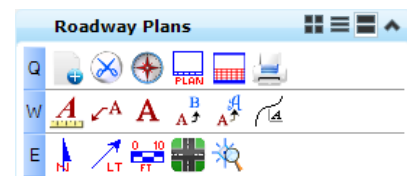
5. Right-click on the Roadway Plans task bar and select **Open 'Roadway Plans' as Toolbox**.
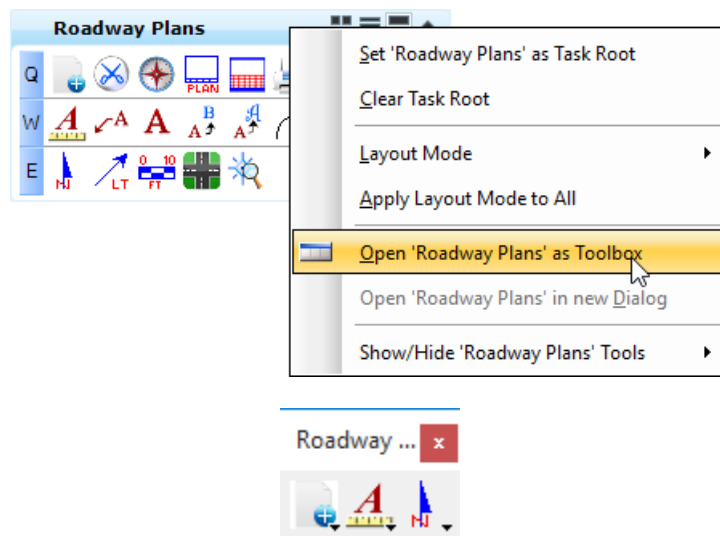
6. Select the **Tasks** tab. The Tasks dialog displays.

7. Click the down arrow to the right on the Tasks tab and review the Task Navigator drop down list.

8. Click the plus (+) icon next to Main Classic task to expand the toolboxes, and then select **Fence**.

9. The Tasks dialog tab changes to Fence tab.

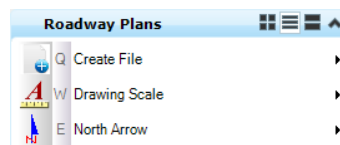10. From the *Task Navigator* dropdown, select **Tasks** to return to the Tasks dialog.

➢ *Toolboxes*

1. From the MicroStation menu, select **Tools** and hover the cursor over the tools in the list to review the popup menus of respective tools.

2. Hover over *Text* and select **Place Text** <OR> key-in the number '**1**' to open the Text Editor window.
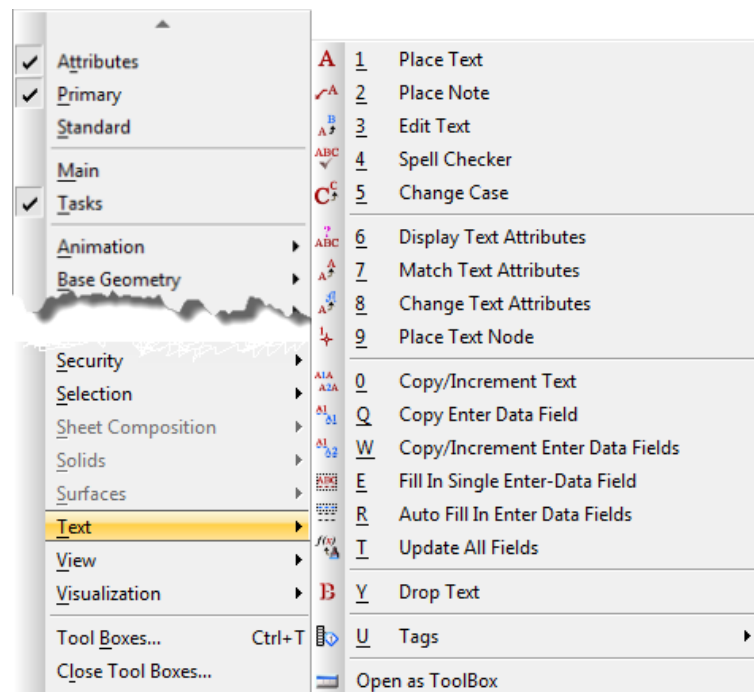
3. From the Text Editor window, select the Close ❌ icon.

4. From the MicroStation menu, select **Tools > Tool Boxes** to review all Toolboxes, Tool Frames and Tasks available for the user to select and open. The Tool Boxes dialog displays.

5. Select the down arrow on the right side of the tool listing and review the different tool listings from the dropdown menu.
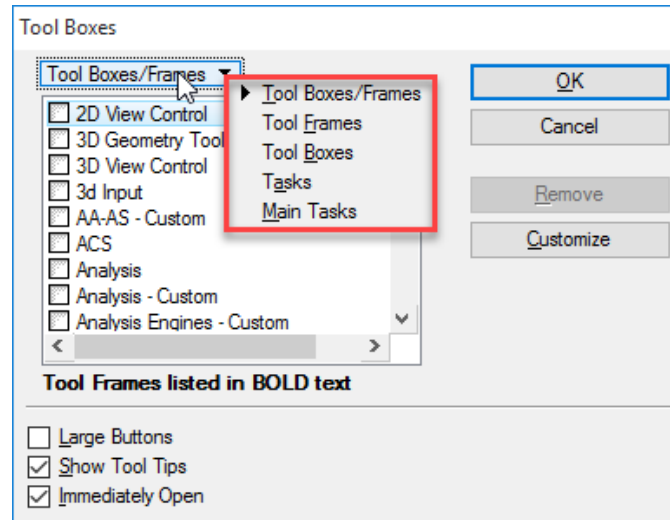
6. From the tool listing window, scroll down and select the **Modify** toolbox and click **OK**. The Modify toolbox displays and the Tool Boxes dialog disappears.

7. Right click on the Modify toolbox to bring up the tool selection popup menu. Notice that only one tool at a time can be checked or unchecked for displaying in toolbox.
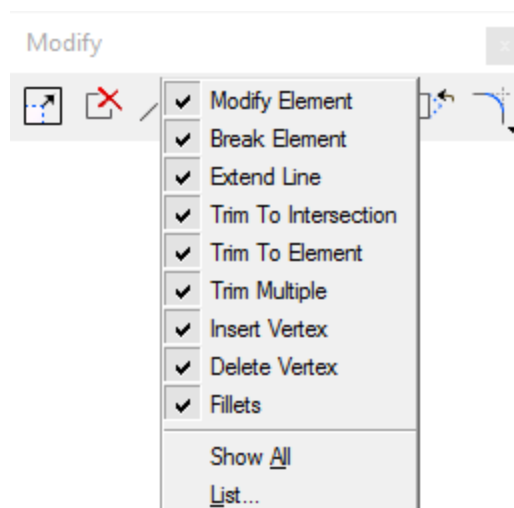
8. From the tool selection popup menu, select List… Show/Hide Tools dialog displays. Notice that multiple selections can be made by checking or un-checking tool options for display in the toolbox.
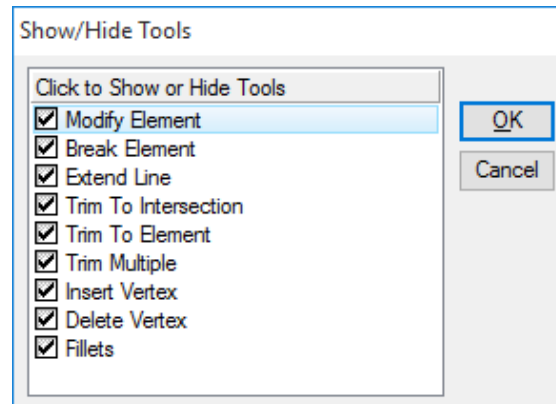
9. Un-check **Trim to Intersection** and **Insert Vertex**, then click **OK**. These tools are removed from the Modify toolbox.

10. Select and hold the title bar of the Modify toolbox and then drag it to the right side of the application window. The toolbox will 'jump' to the side and reduce in size to a tool bar. Release the hold on the title bar. It is now docked to the side of the screen layout.

11. Select and hold at the front dotted bar of the Modify toolbox, and then drag to the center of the drawing area to make it "float" again.

12. To resize the Modify toolbox, select and hold the lower right corner, and then drag to the right of the toolbox until desired size.

Once docked, select dotted bar and then drag to view window.

Select & hold corner, then drag to the desired size.
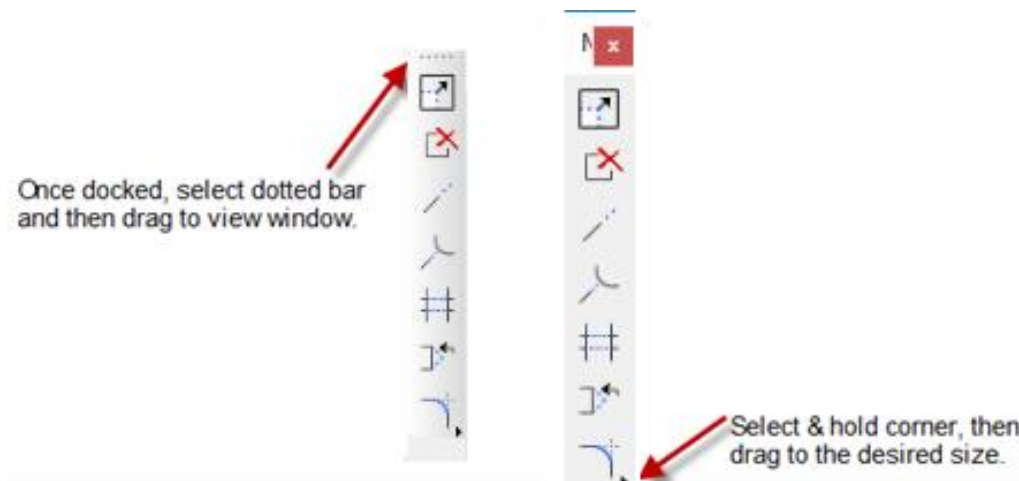
➤ *Tool settings*

1. From the Main Classic task, click on the Place Line. The Place Line tool settings dialog displays.
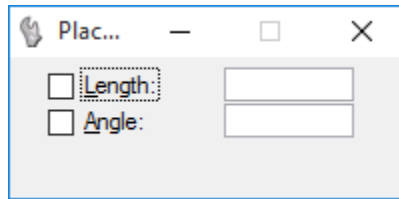
2. From the Main Classic task, click on the arrow on the right side of Place Line and select Place SmartLine from the popup menu. The tool settings dialog changes to accommodate data entry specifically for placing a SmartLine.

3. Click the down arrow in the lower right corner to open additional tool setting options.
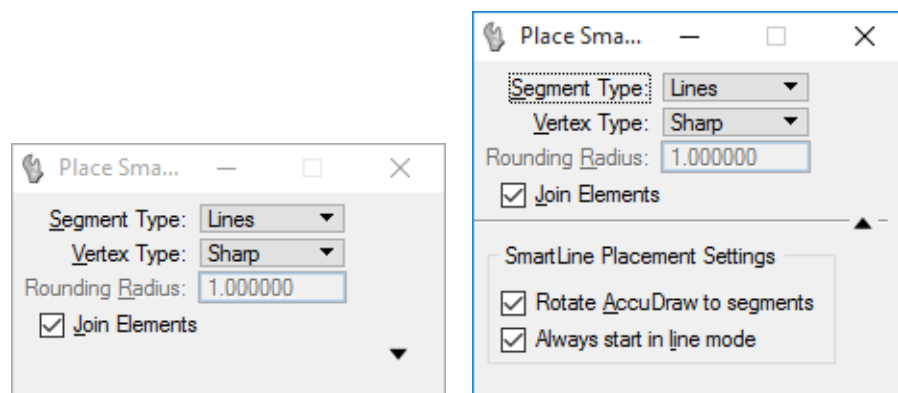
4. Press Esc key and reveal the Keyboard Input for the tool settings options.

5. Click OK, then ✖ to close dialog.

➤ *PopSet*

1. (Optional) If the Primary Tools toolbox is not displayed, select the MicroStation menu option Tools, then check **Primary**. The Primary Tools toolbox will display at the top.

2. (Optional) If the Primary Tools toolbox is displayed, but the PopSet icon is not included, right click on the **Primary Tools** toolbox and check **PopSet**. This will display the PopSet icon.

3. Right click on the **Enable/Disable PopSet** icon and select **Properties**.

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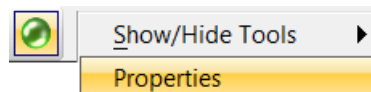
4. Check ON the Auto-Position *Tool Settings* and click OK.

PopSet Settings

☑ Auto-Position Tool Settings
☐ Allow Ctrl+Space When Disabled
New Tool Delay (seconds): 0.75
Hide Delay (seconds): 0.25
Hide Border (pixels): 15

Focus Item Description

Option to have PopSet auto-position the Tool Settings
when the pointer is over a selected icon. If not set the
Tool Settings can only be shown with Ctrl+Space, by
moving the pointer over the PopSet dialog, or if a non-zero

[OK]   [Cancel]

5. Move the cursor over an active (highlighted) tool icon and notice the tool settings dialog displays next to the icon. Move the cursor away and the Tool Settings disappears.

6. Move the cursor over the **Enable/Disable PopSet** icon and notice the tool settings dialog displays under this icon. Move the cursor away and the Tool Settings disappears.

7. Right click on the **Enable/Disable PopSet** icon and select **Properties**. Review the dialog for additional settings for PopSet and then click **OK**.

➢ *Quick Select Dialogs*

1. Locate the Primary Tools toolbox and drag it to the middle of the screen.

Primary Tools   ☒
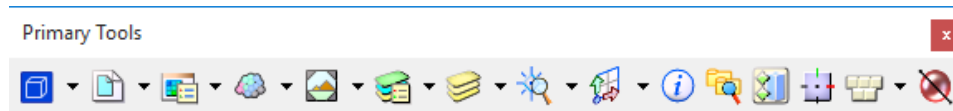
2. Notice that for many of the icons, there is a dropdown arrow beside it. Select these to make dialog settings available without actually "floating" the dialog.

Note   Notice that most of the options are dockable dialogs.

3. Right-click on the toolbox and select **Show All** and review each of the options available.

4. Select the arrow beside the **Level Display** icon.
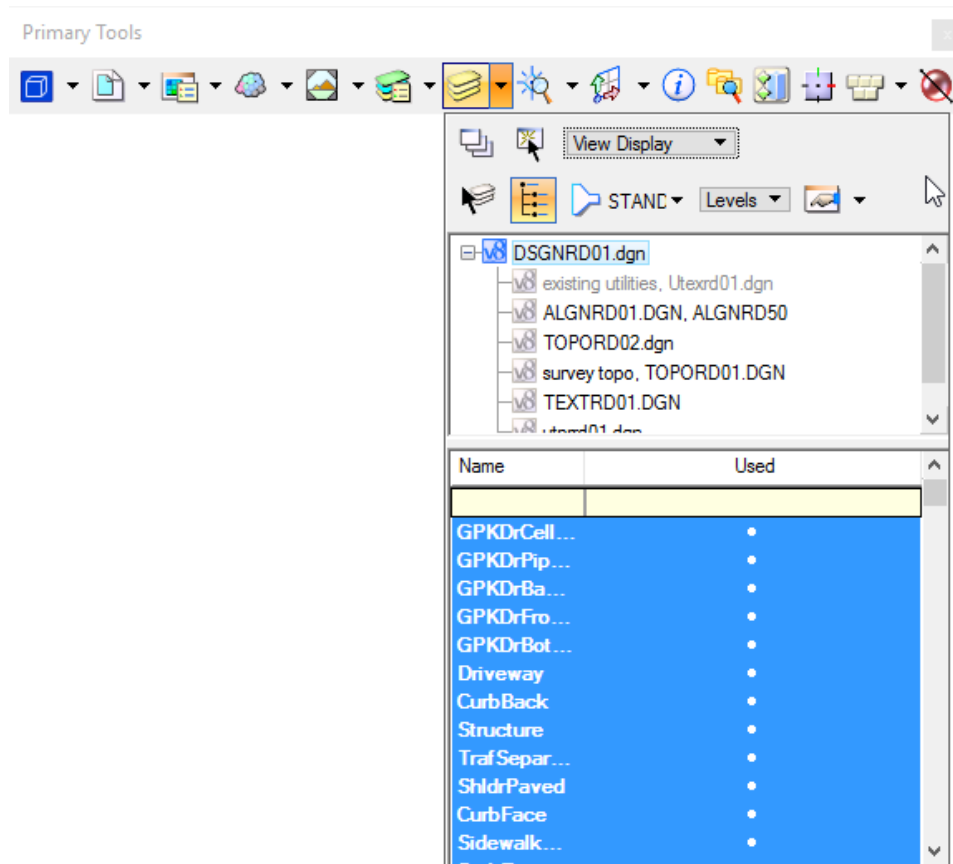
5. From the Level Display, double-click the level named **EOP** to set as the *Active Level.* The EOP level highlights in the level listing, the elements flash within the view window and the Active Level option on the Attributes toolbox at the top of the MicroStation application window changes.

6. From the *Primary Tools* toolbox, click the **Level Display** icon instead of the arrow. Now the Level Display dialog opens as a floating dialog and can be accessed until it is no longer needed.

Note   Notice that there is a FDOT Standards Filter (DSGNRD) applied to the design file. This only displays valid levels within this specific design file.

7. To see all levels used within the file, select the down arrow on the *List Filter* and select **NONE**.
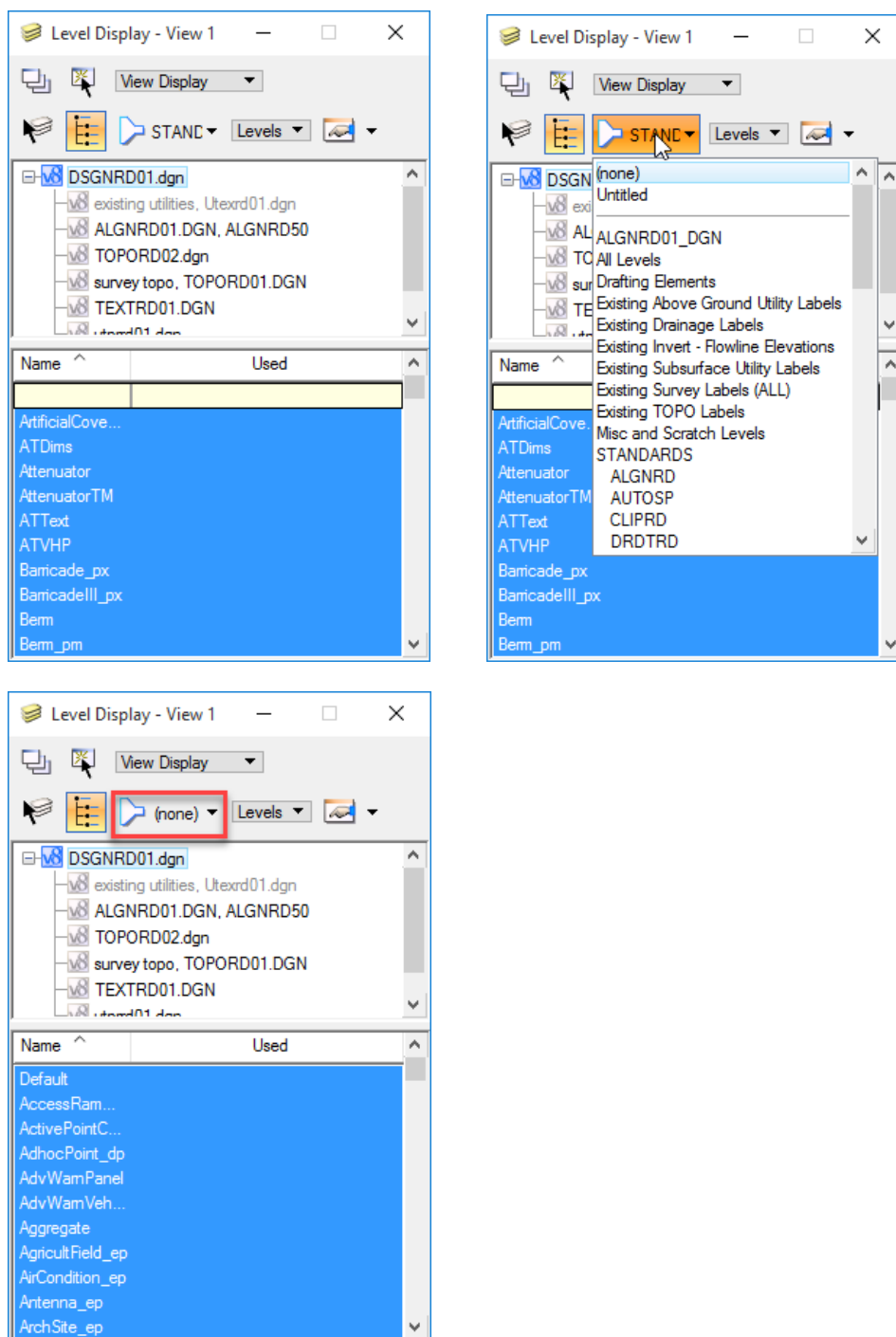
8. Click the **Close** icon to close dialog.

9. Select and hold the Primary Tools title bar and drag to the top of the MicroStation application window to re-dock the toolbox.

Exercise 2.3 *Dockable Dialogs*

1. Continue with *dsgnrd01.dgn* for this exercise.

2. From the Main Classic task, select **Copy** from the *Manipulate Classic* toolbox. The Copy Element dialog displays.

3. Select and drag the Copy Element Tool Settings by the title bar across each of the docking indicators displayed in the window. Notice a transparent box 'jumps' to the position indicated by the arrow.
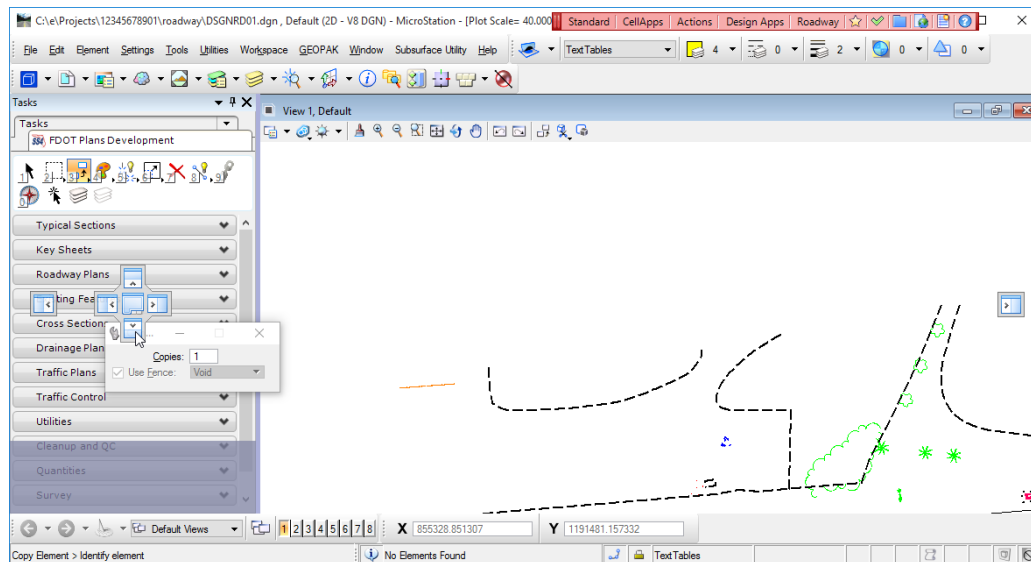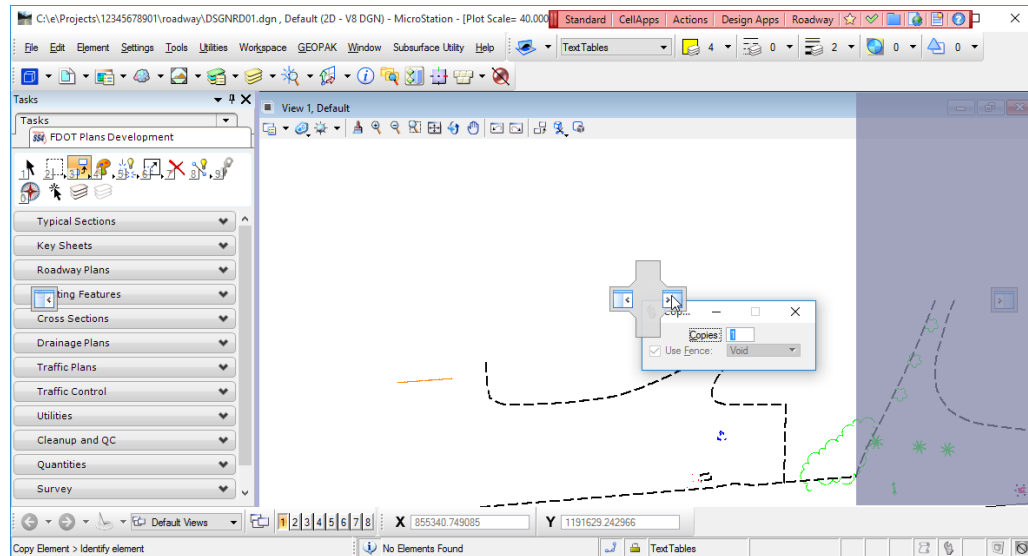
4. Release to the right side. The Copy Element dialog docks along the side of the view window.
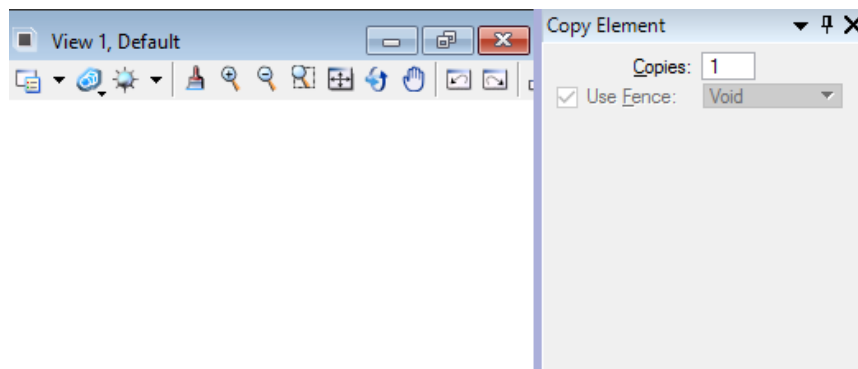
5. From the docked Copy Element dialog, select the Auto Hide (pushpin) icon to minimize the dialog. The Copy Element dialog reduces to a tab. From the docked Task dialog, select the Auto Hide (pushpin) icon to minimize and reduce to a tab.
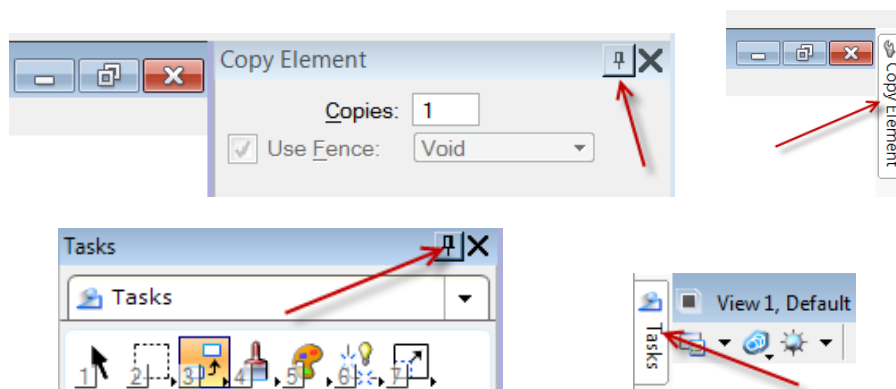
6. From the Primary Tools toolbox, click on **Models** and then click on **References** dialogs. Dock each dialog to the bottom of the view window.
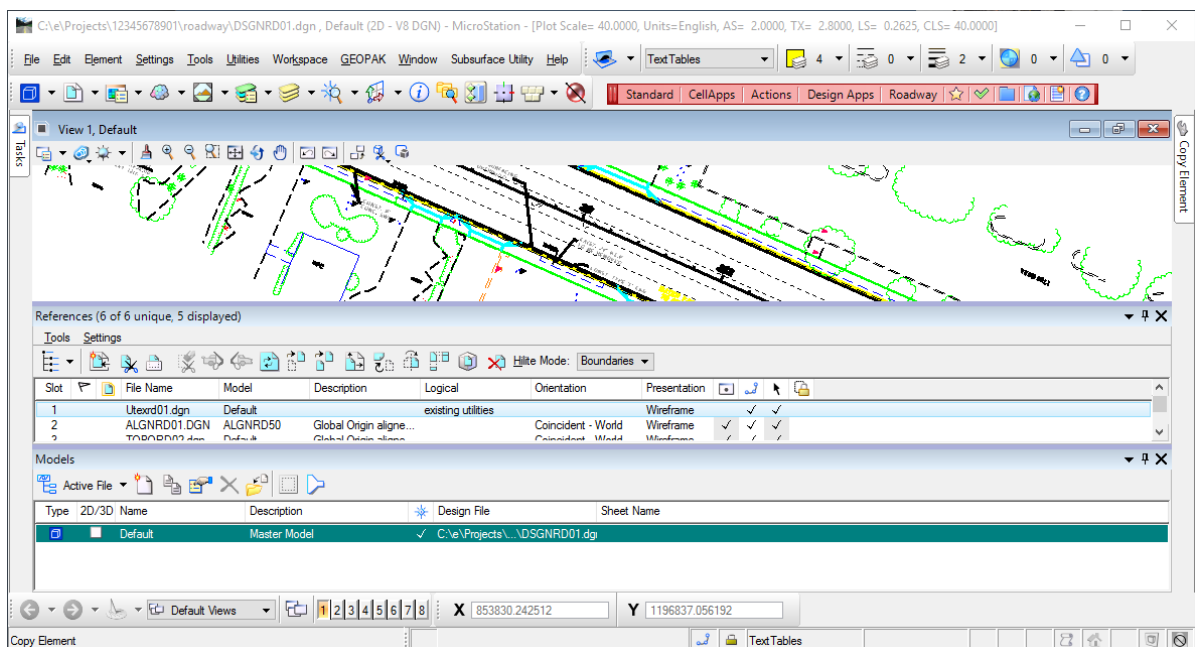
7. On each Models and References dialog title bar, click the **Auto Hide** (pushpin) 📌 icon to minimized to tabs.
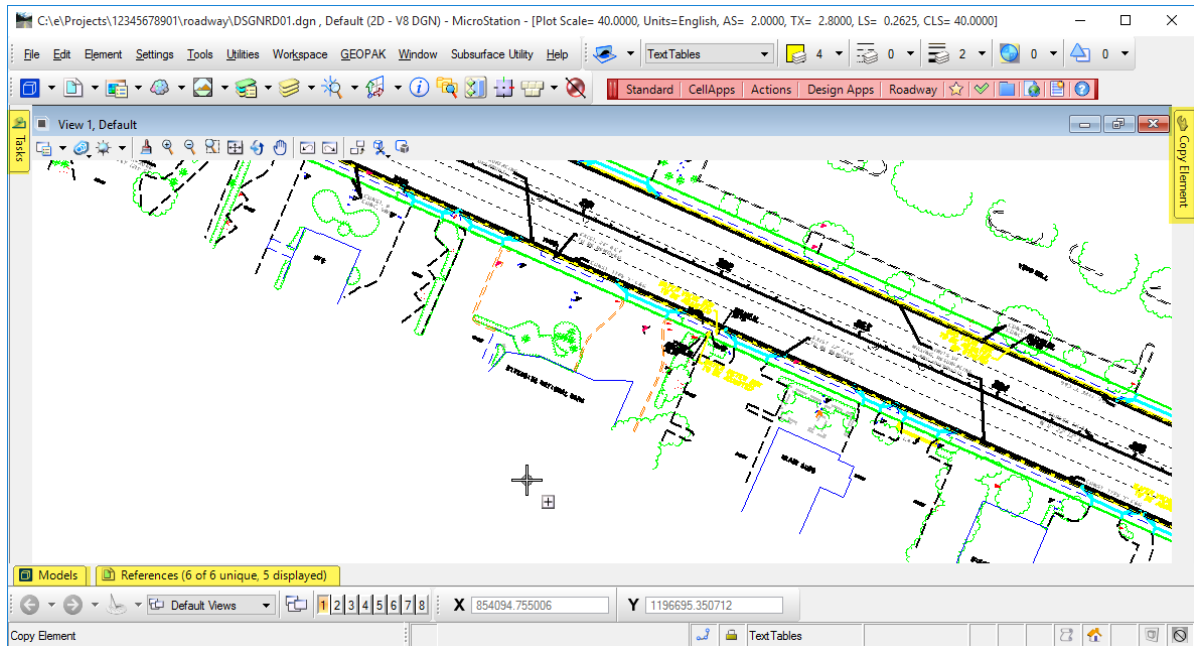
8. To open docked dialogs, make sure the focus is on the MicroStation application window and then **hover** cursor over any tab. The dialog will automatically open for selection.

9. To return dialog to a 'floating' state, open and toggle Auto Hide (pushpin) icon, and then drag dialog from the title bar into the View window.

10. Click the **Close** icon to close each dialog

11. Exit MicroStation.

3   VIEWING & ZOOMING

OBJECTIVES

This chapter covers all the viewing tools available in MicroStation including:

- View Windows
- View Preferences
- View Groups
- View Controls
- Saved Views
- Dynamic Views

INTRODUCTION

As a drawing becomes more and more complex, it may be useful to open multiple view windows. MicroStation can have as many as eight drawing view windows open at one time. Opening multiple view windows allows users to focus on details, as well as an overview of the design, without getting lost in between. The view currently being worked in is known as the *Active View* and indicated by a highlighted title bar.

VIEW WINDOWS

A *View* window in MicroStation is commonly referred to as a "view". Each view functions independently without affecting other views. They contain their own attribute settings with independent zoom, pan and rotate capabilities. Each can be resized or relocated independently by "clicking and dragging" the cursor at any edge of the view or on the title bar. View Windows can be arranged manually or by automated tools found under the MicroStation menu option **Window**, as shown below.
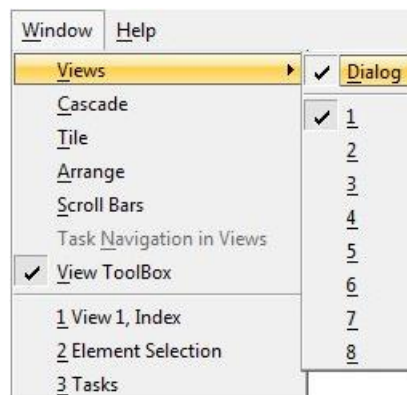
Views	Opens or closes each view window by checking or un-checking the *<view number>* respectively and opens or closes a floating or dockable View Groups dialog as described in the next section.
Cascade	Layers the views on top each of each other.
Tile	Tiles the views to the best fit.

Arrange	Fits the open views together according to their current position on the screen.
Scroll Bars	Adds or Removes the scroll bars along the side and bottom of the view windows.
Task Navigation in Views	Toggle to provide the Task Navigation tool within each view window.
View Toolbox	Toggle to provide the View Control tool bar within each view window as described in a separate section below.

Note   View Options under the Preferences dialog consists of controls for settings to customize the MicroStation view window look/layout. To access the Preferences dialog, select the MicroStation menu option Workspace > Preferences.

VIEW PREFERENCES

The workspace Preferences dialog contains controls for settings to customize MicroStation's view window look/layout.
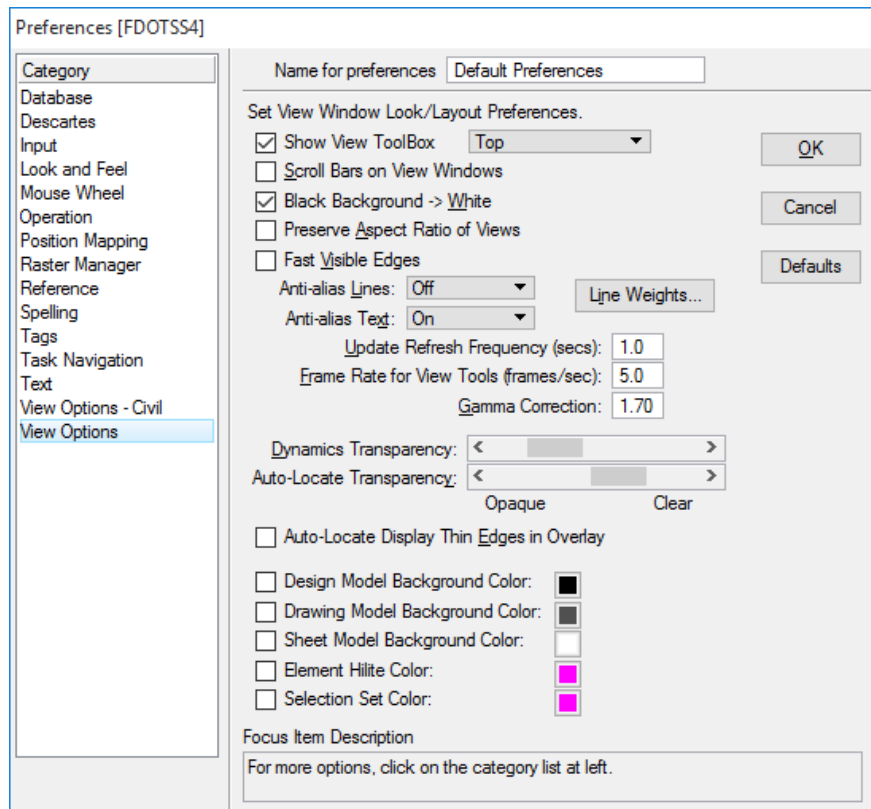
Show View ToolBox	If on, sets the position of where to dock the tools in a view window. The default is top, but can be docked to the left, right or bottom as well.
Scroll Bars on View Windows	If checked on, displays the view windows with borders, including scroll bars and view control bars. The default is off.
Black Background -> White	If checked on, (if set to black) is displayed in white. The default is off.

       FDOT MicroStation Essentials - Part I

VIEW GROUPS

A *View Group* is a named collection of eight view windows which allows for the setup of the desktop to display preferences including number of open view windows, window size and view orientation. Each view group is associated with a model making it easy to access and navigate through different models in the active DGN file. The View Groups dialog provides for this navigational capability and is accessed through the MicroStation menu option **Window > Views > Dialog**.
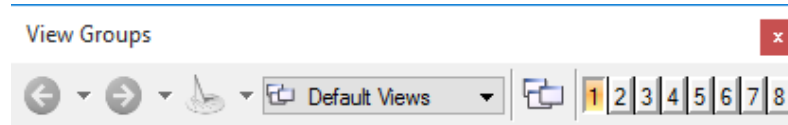
Previous Model   Accesses the previous model in the session history by clicking the left arrow. Alternatively, clicking the down arrow and selecting a previously accessed model from a drop-down list.

Next Model   Accesses the next model in the session history by clicking the right arrow. Alternatively, clicking the down arrow and selecting the next model from a drop-down list.

All Models Visited   Clicking the down arrow displays a drop-down list of all models accessed, from which a model can be selected to open.

View Groups   Drop-down list that displays the name of the view group as well as the model with which it is associated and allows for the selection of different view groups.
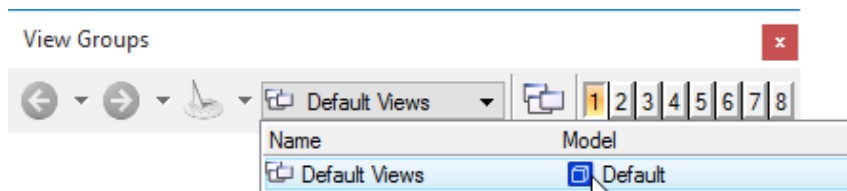
Manage View Groups   Creates, manages, edits and deletes view groups. MicroStation allows an unlimited number of view groups to be saved. When users create a model, they are prompted to create a view group associated with that model by populating the *Create a View Group* checkbox in the Create Model dialog.
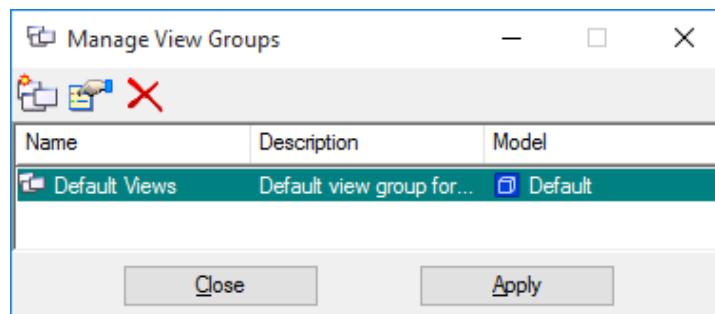
Create View Group   Opens the Create View Group dialog to specify a name and description for the new view group.

Edit View Group Properties   Opens the View Group Properties dialog to edit the name and description of the selected view group.

|	
Delete View Group	Deletes the currently selected view group.
List Box	Displays a complete list of all view groups for all models in the design file. The active view group is highlighted in green.
Apply	Applies the selected view group to MicroStation, opens the views in that view group and activates the model associated with the view group.
Close	Closes the Manage View Groups dialog.
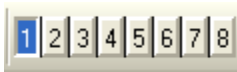
Toggles   Click the numbered buttons to open or close individual view windows (equivalent to choosing the numbered items in the Window menu's Views submenu.) The numbered button grouping is also present in various other dialogs, such as the References and the Level Manager dialog (Settings > Level > Manager).

VIEW CONTROLS

MicroStation contains a complete set of viewing tools, including tools for zooming, rotating and panning views and fitting designs to a view window. Using the View Control tools is one of the most common tasks in MicroStation.

In the upper left corner of each view window is the View Control tool bar, which contains a row of tools used to perform a variety of viewing and zooming operations. Right clicking on the tool bar, offers the user a popup menu to customize the display of tools by checking or un-checking any tool.
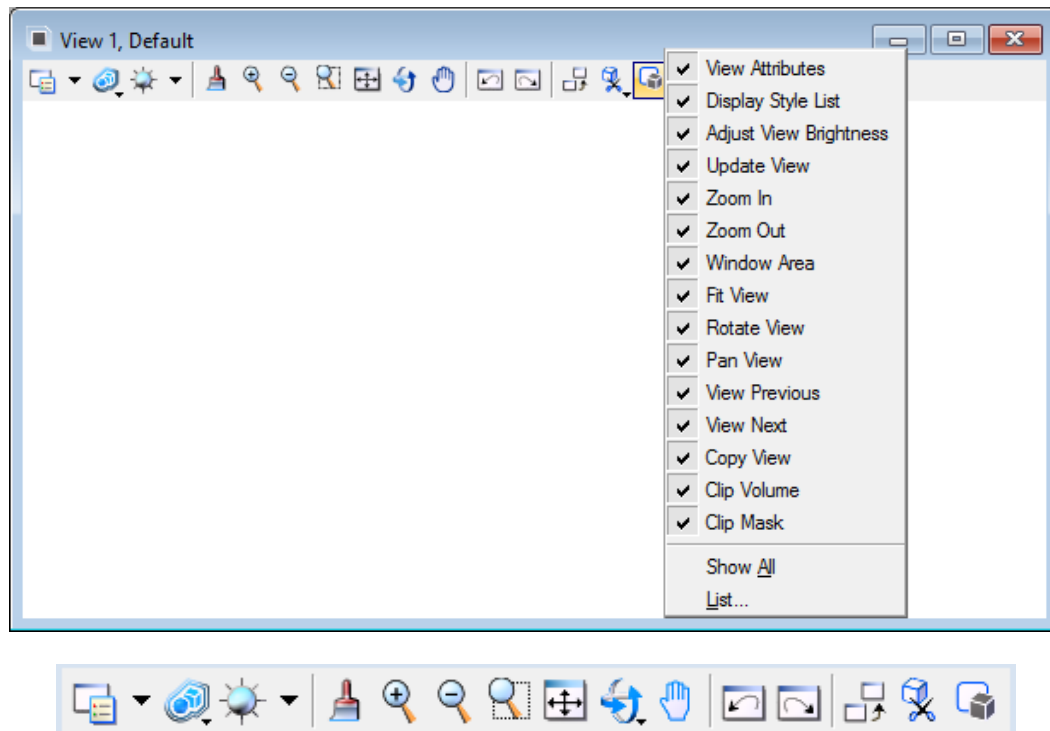
VIEW CONTROL TOOLS

	View Attributes	Opens the View Attributes dialog to adjust view attribute settings.
	View Display Mode	Sets the view display mode from the list of display styles or from the Display Styles dialog.
	Adjust View Brightness	Applies scene lighting to a view and adjust brightness of shaded views.
	Update View	Refreshes (redraws) the current view to display elements correctly.
	Zoom In	Increases the magnification of a design by a user specified Zoom Ratio.
	Zoom Out	Decreases the magnification of a design by a user specified Zoom Ratio.
	Window Area	Magnifies an area of the design by a user defined window.
	Fit View	Display all displayable elements in the active design and attached references in a view.
	View Rotation	Opens the View Rotation toolbox and rotates/un-rotates the view.
	Pan View	Views a different part of the design without changing the view magnification.
	View Previous	Undoes the last viewing operation.
	View Next	Redoes the last undone viewing operation.
	Copy View	Copies content from one view window to another.
	Apply or Modify Clip Volume	Opens the Clip Volume toolbox and applies a clip volume to a view.
	Clip Mask	Applies a clip mask to a view.

The View Control toolbox also provides tools for the 3D design, which are disabled when working in a 2D design. To access these 3D tools, select the MicroStation menu option **Tools > View Control**. This toolbox may be docked anywhere in the screen layout.
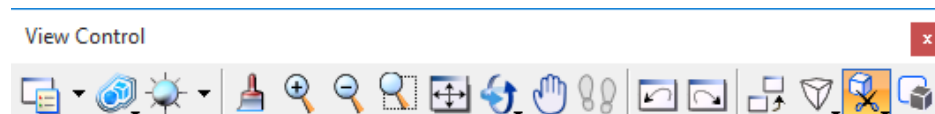
VIEW ATTRIBUTES

The View Attributes dialog contains a series of toggles that allow users to set view attributes that affect whether and how certain types and classes of elements are displayed and whether certain drawing aids are displayed.

Each view can have its own view attributes settings and the view number at the top of the View Attributes dialog determines which view window the settings are applied to when the *Apply to selected view* button is selected. Selecting the *Apply to open views* button applies the current settings to all open view windows.

View Attributes are saved with the design file settings. To save changes to the View Attributes, select the menu option *File > Save Settings*.

The View Attributes dialog can be invoked by one of the following methods:

- Type the key combination **[Ctrl] + [B]**.

- From the MicroStation menu, selecting **Settings > View Attributes**.

- From the View Control toolbox, select the down arrow of the **View Attribute** icon to display a temporary popup of the View Attributes <OR> click on the **View Attribute** icon to open a "floating" **View Attributes** dialog.
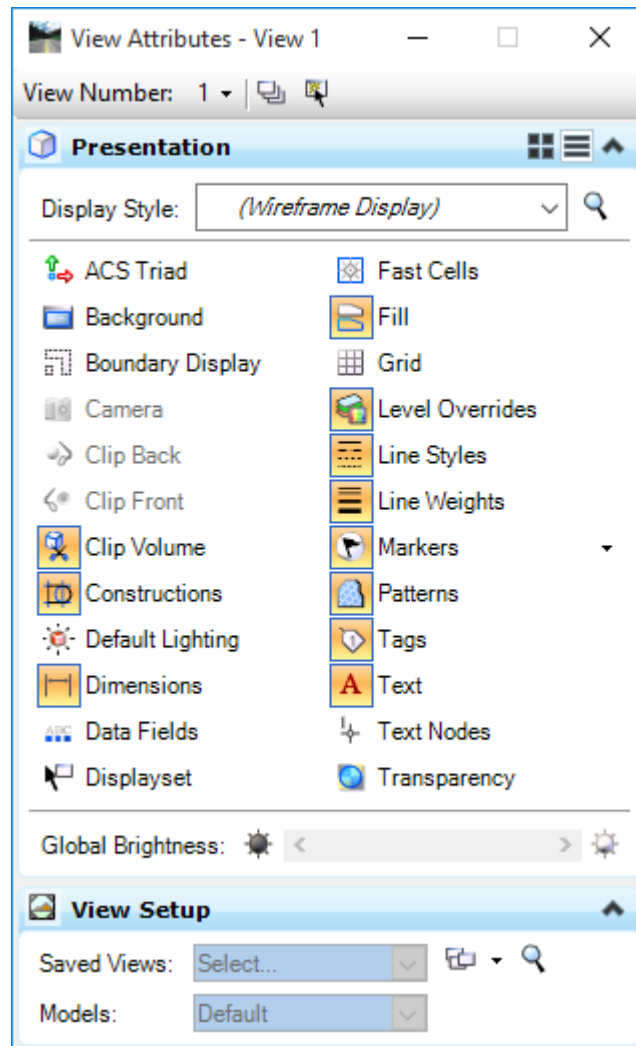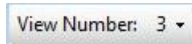
View Number Sets the view for which attribute settings are displayed.

Apply to Open Views If on, any settings changes are applied to all the open views.

Apply to Selected Views If on, any settings changes are applied to selected open views.

Presentation
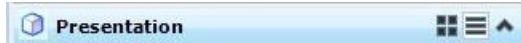
ACS Triad If on, a coordinate triad representing the Active Auxiliary Coordinate System (ACS) displays. The ACS is controlled using the Auxiliary Coordinates dialog, which is opened by choosing Auxiliary Coordinates from the Utilities menu.

Background If on, the background displays.

Boundary Display Controls whether the boundaries of the clip volume are displayed for a given view, as well as reference clip boundaries.

Camera If on, the view camera is on for the view. Applies to 3D designs only. This control is disabled (dimmed) if the active DGN file is 2D.

Clip Back Toggles the display of elements and parts of elements located outside a 3D view's clipping planes. If on, a back clipping plane is active in a view.

Clip Front Enables toggling the display of elements and parts of elements located outside a 3D view's clipping planes. If on, a front clipping plane is active in a view.

Clip Volume Toggles the display of elements and parts of elements located outside a defined Clip Volume for a given view. If on, and a clip volume has been applied to the view, the view volume is restricted to the defined volume. If no clip volume has been applied to the view, it has no effect.

Constructions If on, construction elements (those with the Class attribute of Construction) display. Constructions typically are turned off in a view before plotting or rendering.

Dimensions If on, dimension elements display. Turning Dimensions off can decrease view update time.

Data Fields If on, Enter data fields are displayed.

Display set If on, display of elements is restricted to only those elements in the current display set.

Fast Cells If on, cells display as outlines. Turning Fast Cells on decreases view update time.

Fast Curves If on, curve elements display as line strings (with line segments connecting the vertices). If off, curves are smooth.

Fill If on, solid, closed elements with Fill Type of None or Opaque and text characters in a filled font display with color fill.

Grid If on, the grid displays. Its uniformly placed dots (grid points) and crosses (grid references) provide an indication of distances as a visual aid and a means of facilitating drawing precision with Grid Lock.

Level Overrides	If on, level symbology overrides — the override color, numbered line style, line weight, and material associated with each level — display in the view, instead of the color, numbered line style, and line weight of each element.
Line Styles	If on, elements display with their custom line style, if they have one. If off, all elements with custom line styles display with the standard solid line style (0). Turning Line Styles off can decrease view update time.
Line Weights	If on, elements display with their line weight. If off, all elements display with a line weight of zero. Turning Line Weights off can decrease view update time.
Patterns	If on, pattern elements display. Turning Patterns off can decrease view update time.
Pattern/Bump Maps	If off, pattern and bump maps are not displayed for Phong or Smooth rendering. This can reduce rendering times.
Tags	If on, tags display in the selected view(s).
Text	If on, text elements display. Since text elements take longer to display than most other elements, turning Text off can decrease view update time.
Text Nodes	If on, empty text nodes displayed as small crosses with numeric identifiers. However, in most cases displaying text nodes is unnecessary.
Transparency	If on, applies transparency to elements, references, and levels.

View Setup ⌃
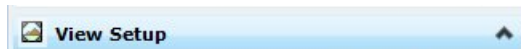
Saved Views	Allows the recall of a saved, predefined view definition to a destination view window and allows use of the selected attributes in the saved view.
Models	Displays the name of the model to which the settings are applied.

SAVED VIEWS

MicroStation allows users to save views for easy retrieval at a later time. A saved view stores the area of the design shown in the view window, view window proportions and location, levels displayed, view attributes and reference settings. The Saved Views dialog can be accessed by selecting **Utilities > Saved Views** from the MicroStation menu bar.
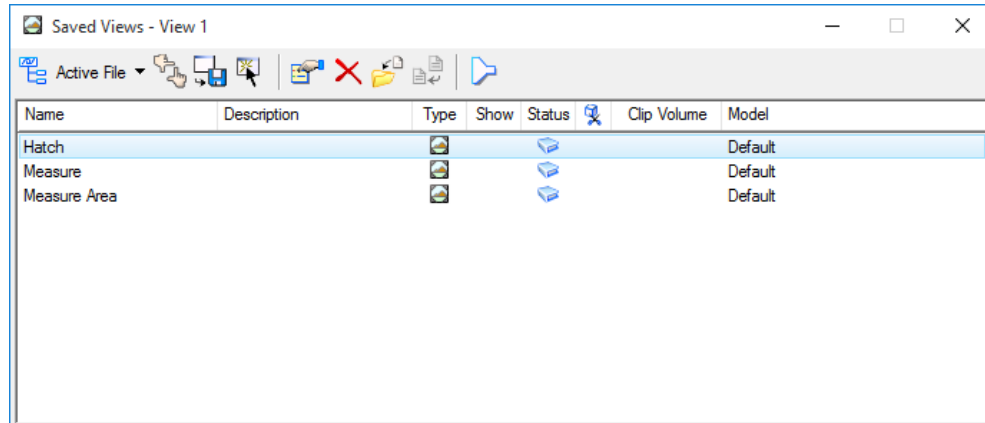
|		
Active File ▾	**Saved Views in Active Design File**	A dropdown list displaying the saved views in the active design file. Click the down arrow and select Active link Sets to see the available link sets in Project Explorer.
	Create Saved View	Opens the Save View dialog, which is used to name and describe the view being saved.
	Update Saved View Settings	Updates a saved view to include changes to the settings. For example, the levels or view attributes can be modified for the saved view, then use this icon to save the changes to the saved view.
	Apply Saved View	The effect is the same as clicking the Apply Saved View toolbox.
	Edit Saved View Properties	Opens the Edit Properties dialog, which is used to change the name and description of the saved view selected from the list box.
✕	**Delete Saved View**	Deletes the selected saved views.
	Import Saved View	Opens the Open File dialog used to select the file that has the saved views to import into the active file. Once a file has been selected, the Import Saved Views dialog opens to import saved views from another file. If no saved views are found to import, an error displays in the status bar.
	Update From Library	Updates the saved view in the DGN file to reflect changes in the configured DGN library.
	List Filter	If on, displays a filter row in the list box that allows for the defining of any column.

DYNAMIC VIEWS

Dynamic Views is a general name that encompasses several related technologies which share a common goal of making model analysis and documentation more interactive and intuitive. One of these technologies allows clipping of models and generating section graphics on the fly. Section views, detail views, and elevation views are types of dynamic views. Dynamic views have the ability to create live, intelligent sections of design composition that updates automatically as the design evolves. More will be discussed in chapters to follow.

Exercise 3.1 *Working with Views*

> ### Zooming and Rotating

1. Open the MicroStation file c:\e\projects\12345678901\roadway*dsgnrd01.dgn* and select the **Fit View** icon.

2. From the MicroStation menu, select **Workspace > Preferences** and then select the *View Options* category.

3. From the Show View toolbox dropdown option, select **Bottom**, and then click **OK**. The *View Controls* will display at the bottom of the View window.

4. From the View Controls located at the bottom of *View 1,* click the **Zoom In** icon. A transparent box will attach to the cursor and the Zoom In tool settings dialog displays.

5. From the *Zoom In* tool settings dialog, set the *Zoom Ratio* to **5.0**, and then zoom in to the last intersection in the bottom right side of the design by using the following methods.

6. Center over the last intersection and click. Repeat until details are displayed as shown below, then right click to accept.

7. Center over the last intersection and scroll with the scroll wheel of the mouse until details are displayed as shown below.
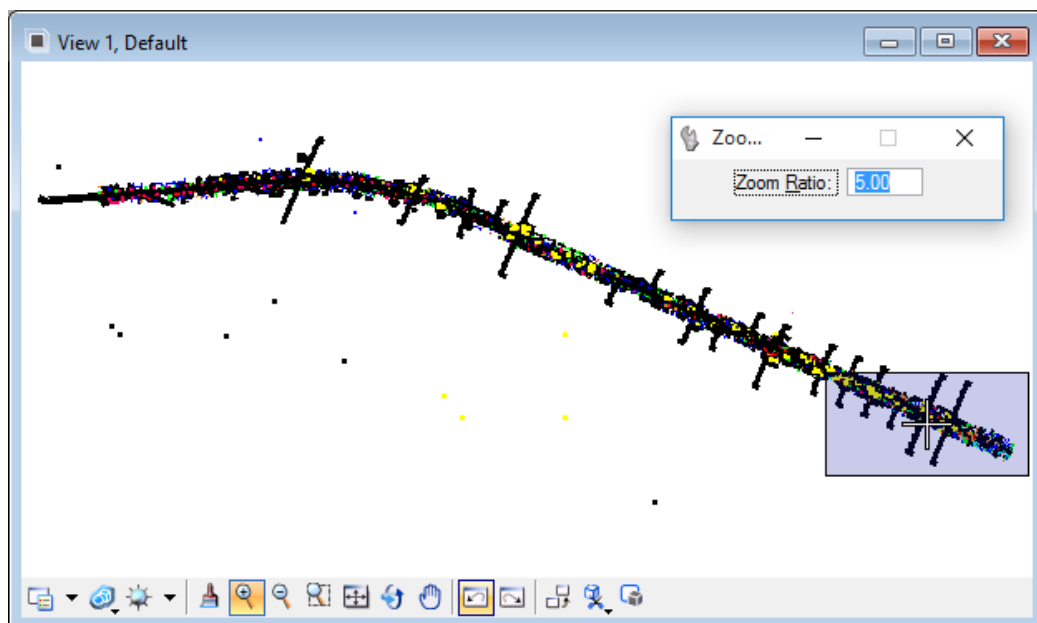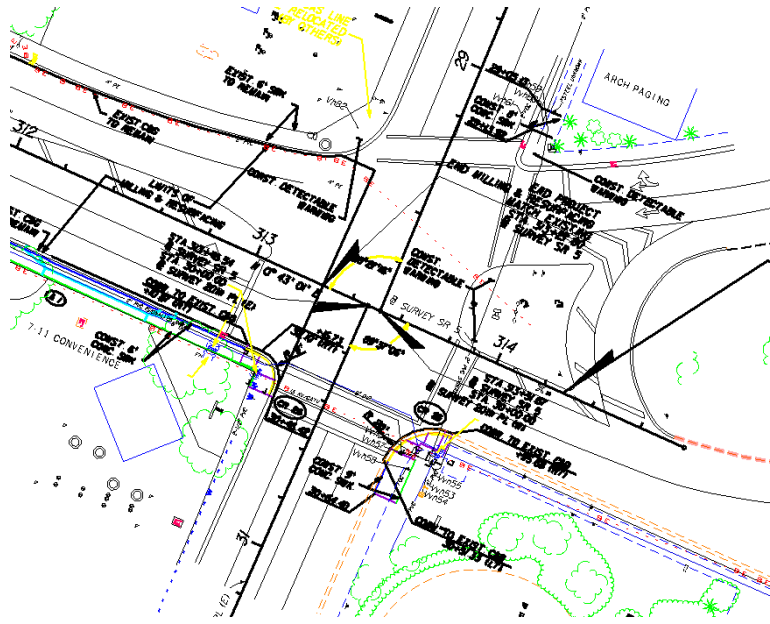
8. From the View Controls, click the **Rotate View** icon. The Rotate View tool settings dialog displays.
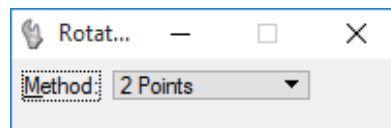
9. From the Rotate View dialog, set the *Method* to **2 Points**. The prompt '*Rotate View > Define first point*' displays in the MicroStation Status Bar lower left hand corner.

10. Define the first point by selecting the intersection of **BL Survey 20th PL (W)** and **BL Survey SR 5**. The prompt '*Rotate View > Define X-axis of view*' displays in the MicroStation Status Bar.

11. Define the X-axis of view (second point) by selecting the end of **BL Survey SR 5**. The image rotates to match the following. The prompt '*Display complete*' displays in the MicroStation Status Bar.
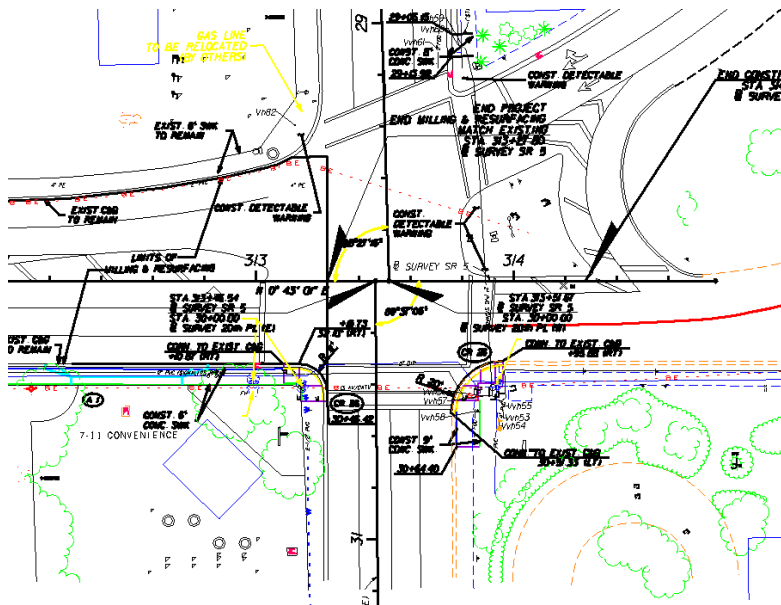
➢ *Arranging Views*

1. From the *View Controls*, click the **Fit View** icon to fit the entire roadway section in the view window.

2. Use the **Window Area** icon to zoom the left most intersection as shown below.
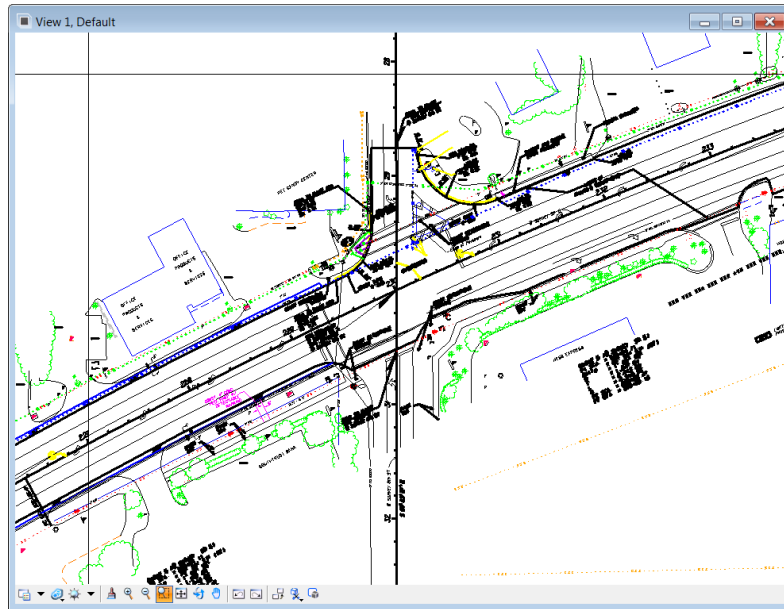
3. Use the **Pan View** icon to pan the view over to the next intersection by picking a point at the left and then choosing a point at the right.

4. From the *View Groups* toolbox docked at the bottom of the MicroStation application window, open a second view window by clicking on the number 2 button.

5. From the MicroStation menu, select the option **Window > Tile** to arrange the Views.
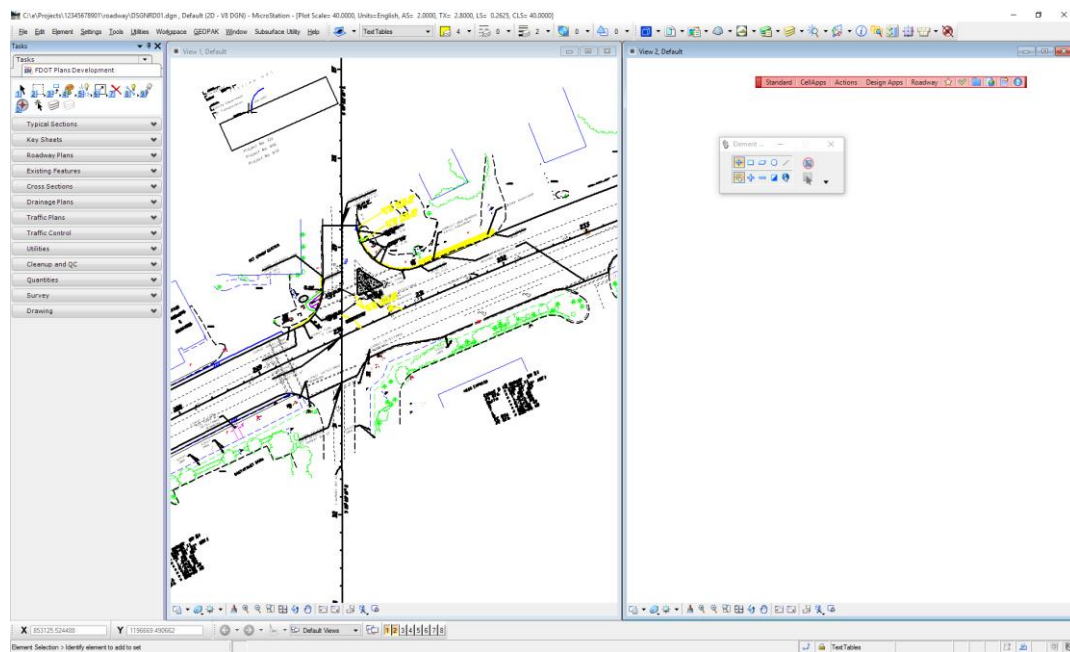
HINT   View 1 should be at the left and View 2 should be at the right.

6.   From the *View Controls* in View 1, click **Fit View**.

7.   From the *View Controls* in View 1, click **Window Area** and change the tool setting *Apply to Window* to **2**.
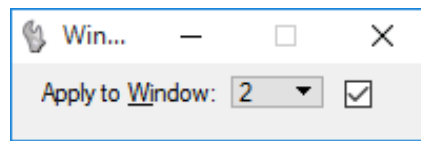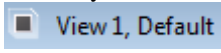
8.   With the *Window Area* tool selected in View 1, window into the far left intersection. The results will be reflected in View 2 instead of View 1. Use **Window Area** several times on different features.
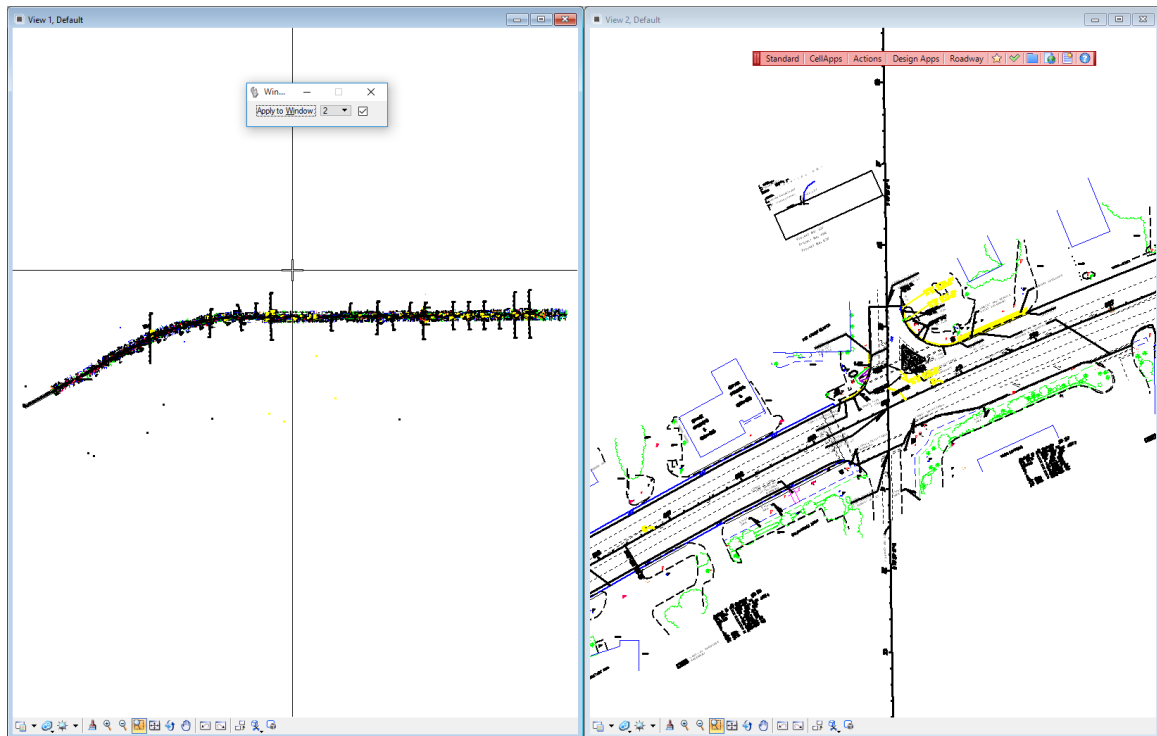
9.   From the View Groups toolbox, close View 2 by clicking on the number **2** button.

10. Expand View 1 to fill the entire work area by double-clicking on the Title Bar.

➢ **_Create and Apply Saved Views_**

1. From the View Controls, click the **Window Area** 🔍 icon to zoom in on the first intersection from the left. This sets the source for the first Saved View.

2. Open the Saved Views dialog by selecting one of the following:

    a. From the MicroStation menu, **Utilities > Saved Views** <OR>

    b. From any view window, click the ■ icon in the upper left hand corner of the view window
       ■ View 1, Default and then select **View Save/Recall** from the popup menu <OR>

    c. In the Primary Tools toolbox, click **Saved View** <OR>

    d. Select the default function key <F6>
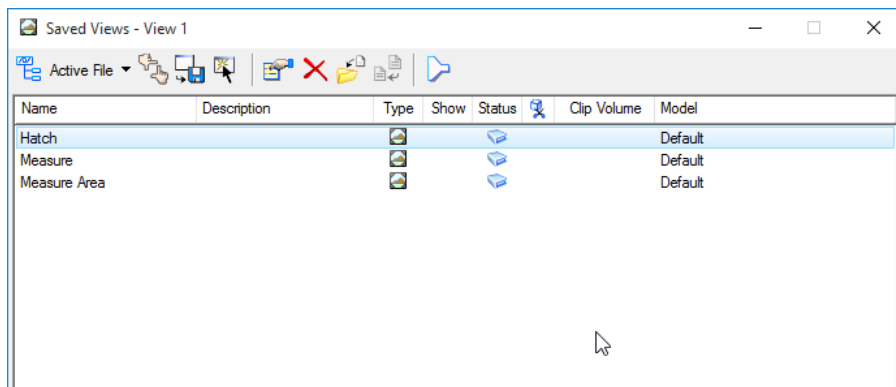
3. From the Saved Views dialog, select the **Create Saved View** 🗔 icon and set each setting as shown below. The Create Saved View dialog displays.

    a. Accept *Method* as **From View**.

    b. Accept *View Type* as **Saved View**.

    c. Enter the *Name* as: **First Intersection**.

    d. Leave *Description* **blank**.

    e. Accept *Clip Volume* as **From View**.

    f. Check On Associative Clip Volume

    g. Leave check box unchecked for Create Dynamic View.

4. Click onto the **View 1** window to select the source view. The Saved Views list box will update with the newly created listing.
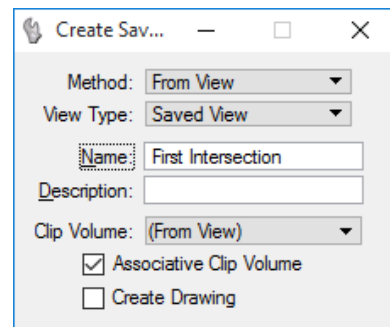
5. Fit View and repeat this process (steps 1-4) to store a saved view for the Second and Third Intersections as shown below.
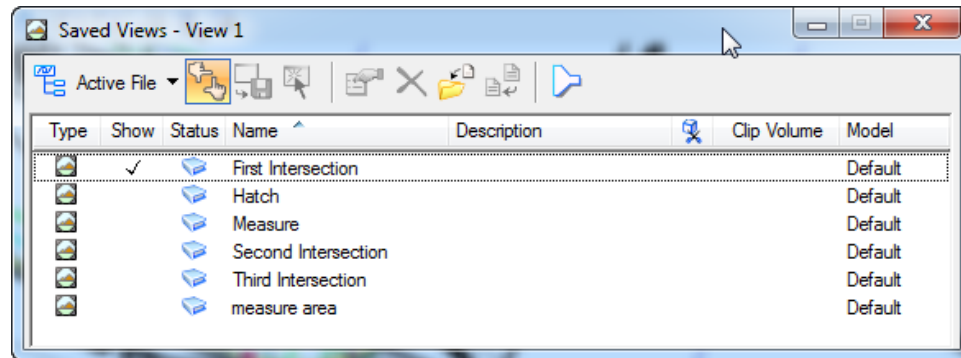
6. From the View Groups toolbox, open **View 2** and **View 3** by clicking the corresponding buttons, and then select the MicroStation menu option **Window > Tile**.

7. From the Saved Views dialog, right-click on the **First Intersection** and select **Apply** from the popup menu. The Apply Saved View dialog displays.
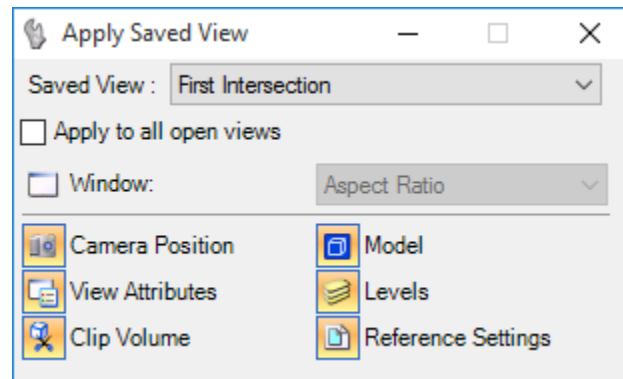
8. Enter a data point in *View 1* to accept the **First Intersection** saved view.

9. From the Apply Saved View dialog, change the *Saved View* option by selecting **Second Intersection** from the dropdown menu, and then enter data point in *View 2*.
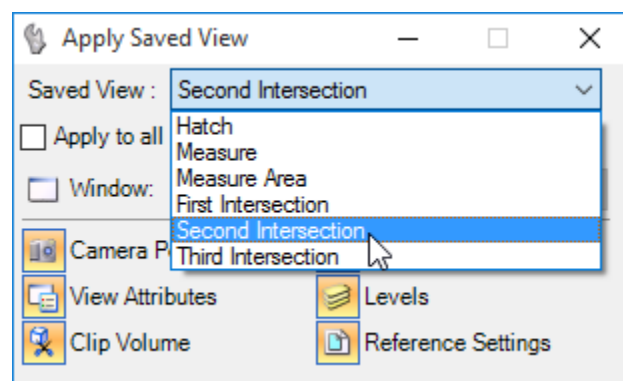
10. Repeat step 9 for the Third Inte**rsection** to View 3.

11. From the View Groups toolbox, close *View 2* and *View 3* by clicking the numbers **2** and **3**, and then maximize and fit *View 1*.

12. Close the Save Views dialog.

➤ *View Groups*

1. From the View Groups dialog, open an additional view such as *View 2*.

2. First, arrange the views as shown below. Then, using the zoom control to arrange the views similar to the illustration below with one view smaller showing more detail and the other view depicting with a larger view.
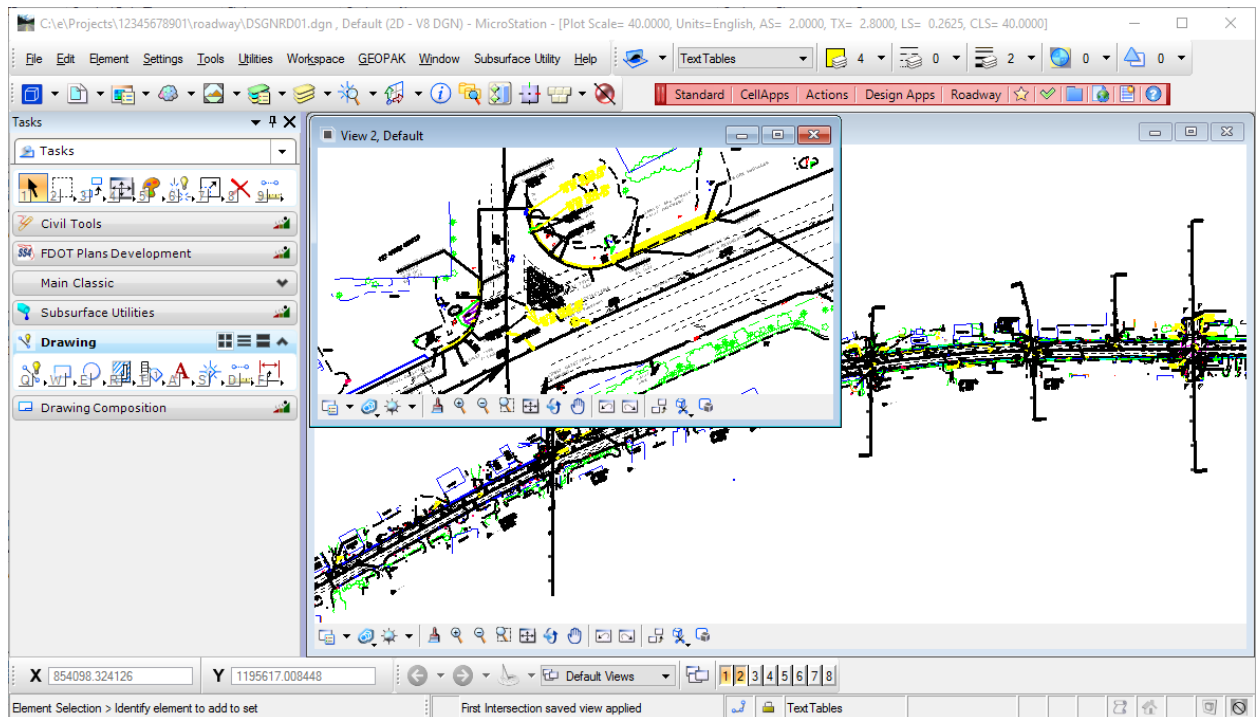
3. From the *View Groups* toolbox, select the **Manage View Group** icon to access the Manage View Groups dialog.
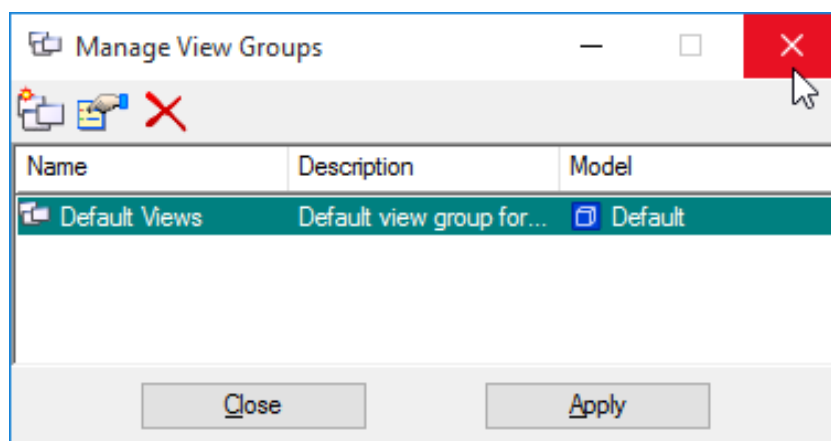
4. From the Manage View Groups dialog, select the **Create View Group** icon to create a new view group.

5. Enter the *Name* as **Detail**.

6. Enter the *Description* as 8th Str and SR5

7. Click **OK**. The *Manage View Groups* list box updates with the newly created View Group.

8. From the Manage View Groups dialog, highlight **Detail** in the listing, enter a data point in the view created, and then click **Apply** to set the view with the newly created **Detail View Group**.
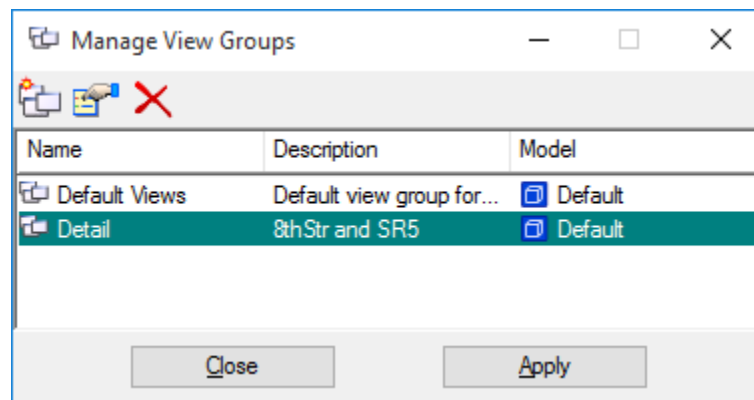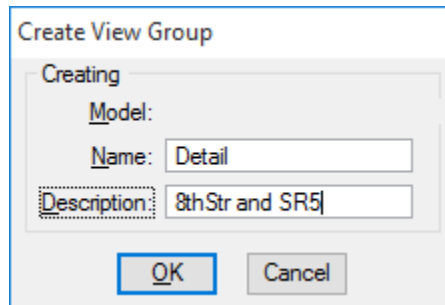
9. Switch between View Groups by double-clicking on **Default Views** to fit the design and close *View 2*. Double-click on **Detail** and see how the view returns to the way it was saved.

10. Close all dialogs and exit MicroStation.

4   MODELS

OBJECTIVES

This chapter focuses on the use of Models and incorporating them into the FDOT environment. Topics include:

- Model Properties
- FDOT's Use of Models
- Models Dialog
- Creating a Model

INTRODUCTION

MicroStation offers many enhancements designed to facilitate the production of engineering information. One of the most beneficial enhancements to MicroStation is the introduction of *Models* in the design file environment.

A design file is composed of models. When users draw or place elements with MicroStation tools, the elements are added to the active model. A model can be either 2D or 3D, and is stored within the design file. When a design file is created from a seed file, an empty container setup is provided with a default model ready to create a design. If a 2D seed file is used, then the default setup is 2D, while a 3D seed file defaults to a 3D setup.

In 2D drafting, the MicroStation equivalent of a sheet of drafting paper is the design plane. Unlike a sheet of drafting paper, the design plane in a design file is extremely large and allows the user to draw models at full scale. To draw various elements in a model, the user enters *data points*. Each data point placed in the design plane has associated X (horizontal) and Y (vertical) positions or coordinates. The design plane is simply a Cartesian coordinate system upon which a model resides.

MODEL PROPERTIES

Each model also contains it own unique set of working units which theoretically would allow metric and English coordinate systems within the same design file. Every model has its own set of eight views. The model whose views are displayed or available for display at a given time is the active model.

There are three types of models: *Design Model, Sheet Model* and *Drawing Model.*

Design Models are similar to DWG models (model space), which consists of design geometry and can be either 2D or 3D. A Design Model can also be used as a reference or placed as a cell. In MicroStation, an unlimited number of design models can be created in a design file. By default the view windows of these models have black backgrounds.

Sheet Models are similar to DWG drawing layouts (paper space), and are commonly used to compose finished and annotated printable drawings. MicroStation allows an unlimited number of sheet models to be created in a design file. By default, the view windows of these models have white backgrounds.

Drawing Models are a subset of a 2D or 3D design model, used to apply annotations, dimensions, callouts, and other embellishments to a design. By default, the view windows of these models have gray backgrounds.

FDOT'S USE OF MODELS

FDOT has implemented the use of models for combining dependent data into a single MicroStation design file on a specific limited basis. FDOT has chosen to use the Design Model for model creation. One example of the use of models in the FDOT Workspace is with FDOT cross section files. The FDOT cross section seed file (*fdotseedxs.dgn*) is setup with four distinct models. Once a cross section file is opened the models are readily available via the Models dialog.
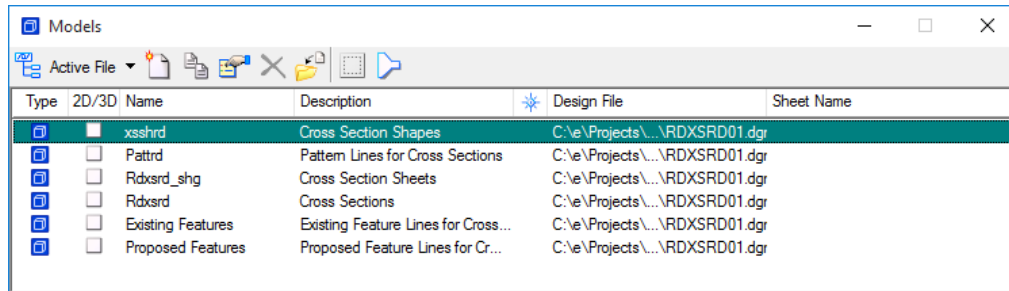
View Groups are predefined for each model within the cross section file. These are created automatically in conjunction with the models and allow easy access to and switching between models.
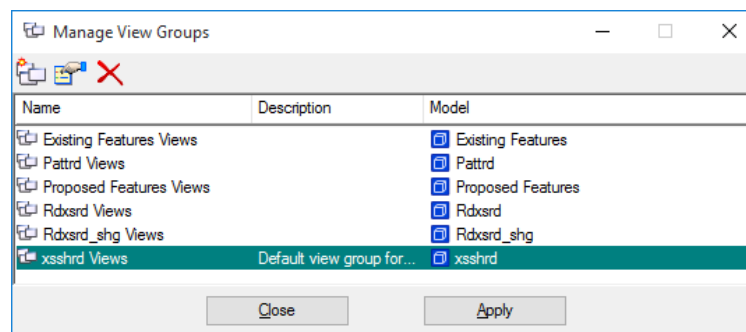
Another example of the use of models in the FDOT Workspace is for alignment stations and tic marks at different scales. This allows the use of one set of level names for all scales, and users of these files to only reference the one model with the needed scale.
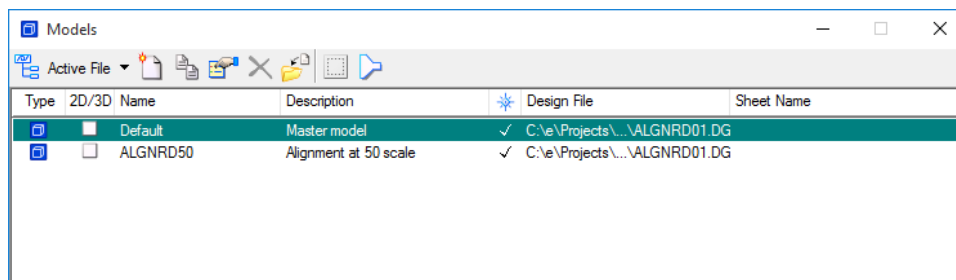
MODELS DIALOG

When a design file is first created, it has a built-in Default design model described as Master Model ready to place elements. MicroStation's Models dialog allows users to create, edit, manage, and quickly switch between any model (including this Default model) in the design file.

This dialog can be accessed from the MicroStation menu option File > Models <OR>
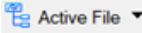
from the Primary Tools toolbox, by clicking on the Models [icon] icon.

Note This dialog can be docked at the top or bottom of the applications window.

The Models dialog contains several tools to allow the user to create, modify and manage models. These tools are outlined in the table below.

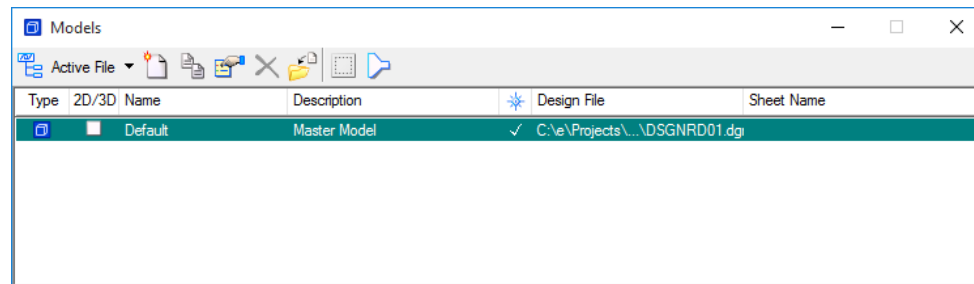|		
Models in Active Design		Displays the models in the active design file. Clicking the down arrow opens a popup menu for selecting an Active Link Tree to see available link sets.
Create a New Model		Opens the Create Model dialog, which is used to create a new Design or Sheet model in the open DGN file. Design models generally contain design elements, while sheet models are used for plotting.
Copy a Model		Opens the Copy Model dialog, which is used to make a copy in the open DGN file of the existing model selected in the list.
Edit Model Properties		Opens the Model Properties dialog, for the model selected in the list box. This dialog is used to modify properties of models contained in the open DGN file.
Delete Models		Deletes the model selected in the list box. The default model is present in every design file and cannot be deleted.
Import Models		Opens the Import Model From File dialog, which is similar to the Open dialog that allows selection of a DGN or DWG file from which to select the model to import. After clicking OK, the Select Models dialog opens allowing selection of the model to import into the open DGN file.
Define Sheet Layout		(Sheet Models only) Opens the Define Sheet Layout dialog, allowing to set the Display status, and define the Size, Origin, and Rotation sheet layout parameters

List Filter
If on, displays a filter row in the list box that allows for defining a filter for any of the list columns.

To define a filter, click on the required filter field and enter the filter parameters. For information on defining filters by string, integer or Boolean expressions, refer to the tables in Defining and Deleting Filters.

List Box
Lists the models in the open DGN file. An icon displays next to the model name indicating whether the model is 2D or 3D. Double-clicking a model here makes it the active model

The model name and description can be edited by triple-clicking on the selected model name in the list box. To copy, delete, or adjust properties of a model, it must first be selected here.

The columns displayed in the list box can be customized by right-clicking in the title row of the list box and using the menu to select which options are displayed.

CREATING A MODEL

Working with *Design Models*, the user can create one or more discrete models within a design files and move between various models using the Models dialog. Where a design file contains many models, filters can be set to display only particular models, but filters are not saved from one design session the next.

The user can create *Sheet Models* to compose design drawings which typically consist of references of the design models. References can be from the open design file or from other design files on disk. As with the Design Models, the user can switch between Sheet Models using the Models dialog.

The *Drawing Models* are the intermediates between the 3D design model and a printable Sheet Model. It is an unbound white space that contains one or more views of the design model. The Drawing Model can be used for centralizing annotations that need to be shared across multiple sheets. When a Drawing Model is attached to a sheet, the drawing model's annotation scale is used as the attachment's default detail scale.

FDOT only supports the use of 2D Design Models within their seed files delivered with the FDOT CADD Software. When creating new models within the FDOT Workspace, the user should set the model *Type* to 2D *Design*.
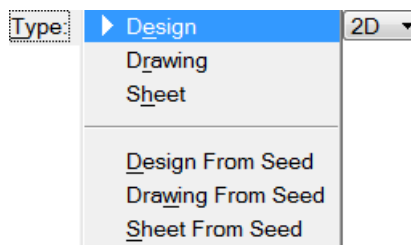
When creating models in a design file, the user has an option to specify whether or not that model "Can be placed as a cell". If this option is turned on, the model can be referenced as a model <OR> placed as a cell within another model. When referencing a model, the link to the original is maintained and any modifications to the original will reflect in the reference. When a model is placed as a cell, the link to the original is removed.

Exercise 4.1   *Models*

> ### Creating Models (Manually)

1. Open the MicroStation file c:\e\projects\12345678901\roadway*algnrd01.dgn*.

2. From the *Primary Tools* toolbox located at the top of the applications window, click the Models icon to open the Models dialog.
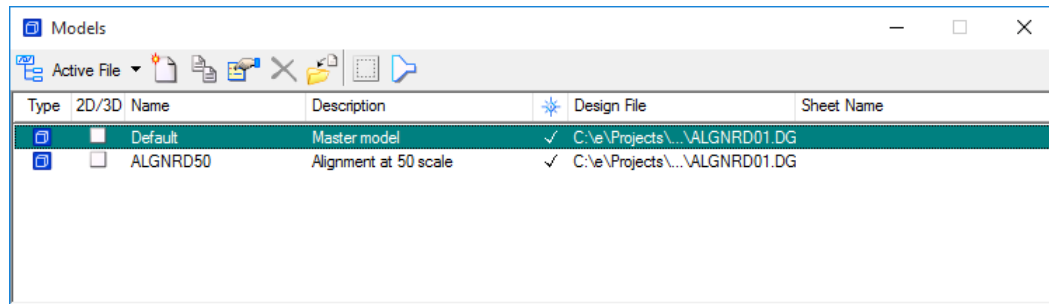
3. From the Models dialog, select the Create a New Model icon to open the Create Model dialog.

4. Complete the Create Model dialog as shown below. Click **OK**. Notice in the Models dialog that the list window updates with the new model named **algnrd40**. At the same time the model was created, MicroStation switched the design environment to the new model for work to begin.
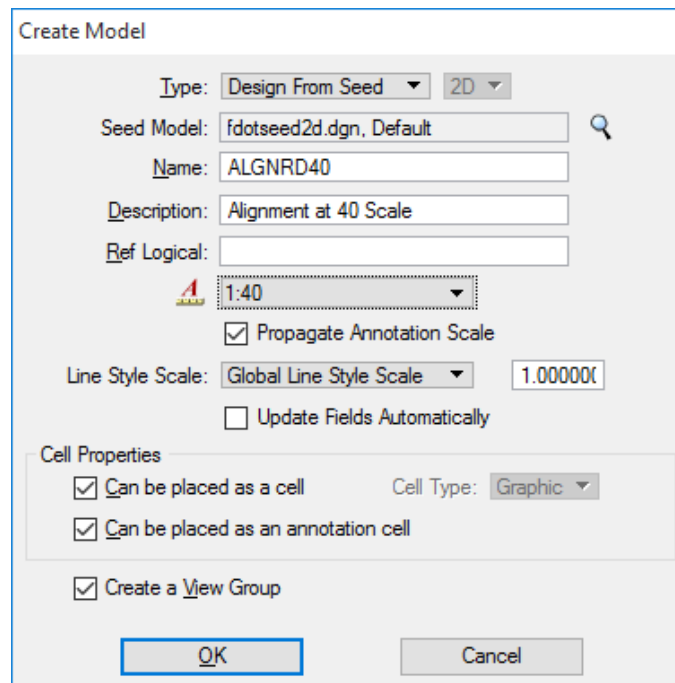
5. Repeat step 4 to create the *Model Name* **algnrd200** with the *Description*: **Alignment at 200 Scale**. Click **OK**.
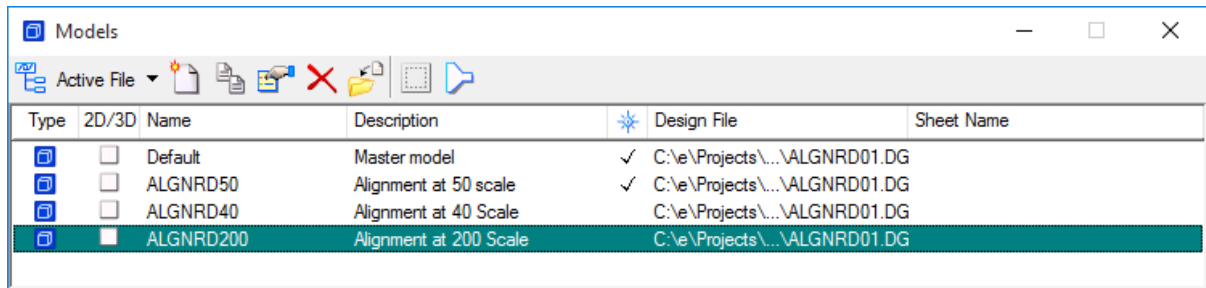
6. From the Models dialog, double-click the **Default** listing to switch back to the **Default** *Model*.

7. From the *View Groups* toolbox located at the bottom of the applications window, notice there is a new corresponding **view group** with names corresponding with each *model* created.
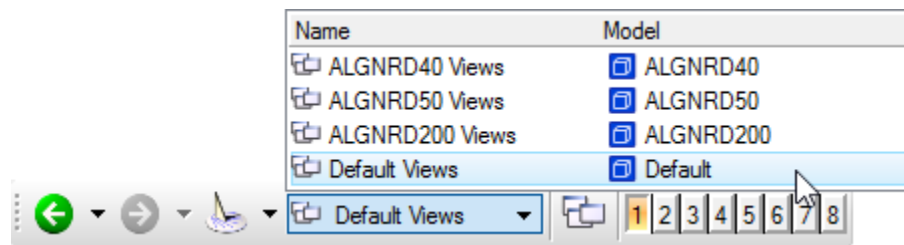
8. Close the Models dialog.

➢ *Creating Models (Automatically)*

FDOT supplies specific preset models within their seed files the use with Cross Section files. This exercise will show how these models are created automatically.

1. From the FDOT Menu, select **Actions > Create Files** to open the Create Files dialog.
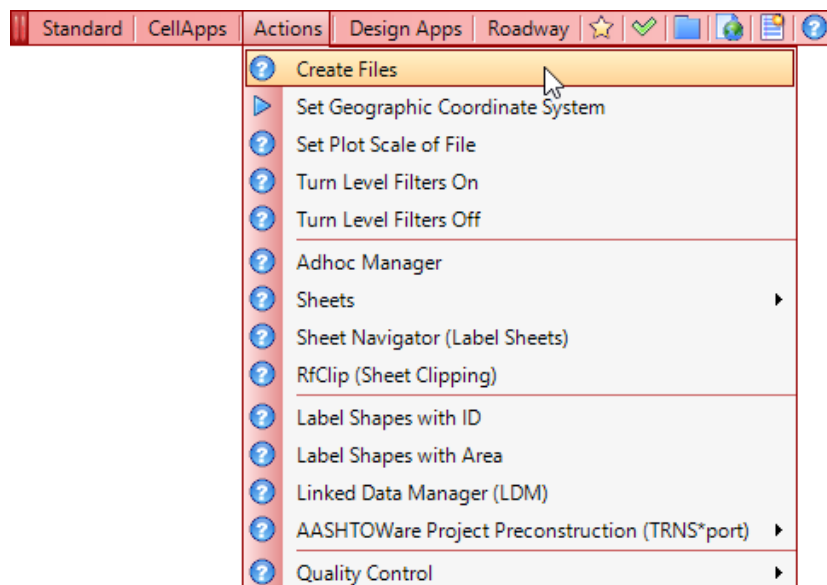
2. From the Create Files dialog, verify/set the following settings as shown below.

    a. *Project* set to **\e\projects\12345678901**. Use the **Project Settings** button, and then click **Select Active Project**. From the Select Active Project, select **Browse** to navigate to this project directory, click **Select** and then click **OK** twice to accept.
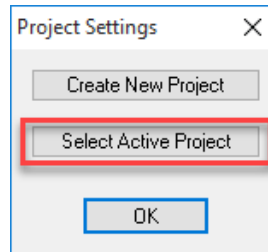
    b. *Workspace* set to **FDOTSS4**.

    c. *Control File* set to **ROADWAY.CTL**.

    d. *File Group* set to **Roadway Design Files (DGN)**.

    e. *File Type* set to **ROADWAY XSECTIONS, PATTERN LINE & SHAPES**.
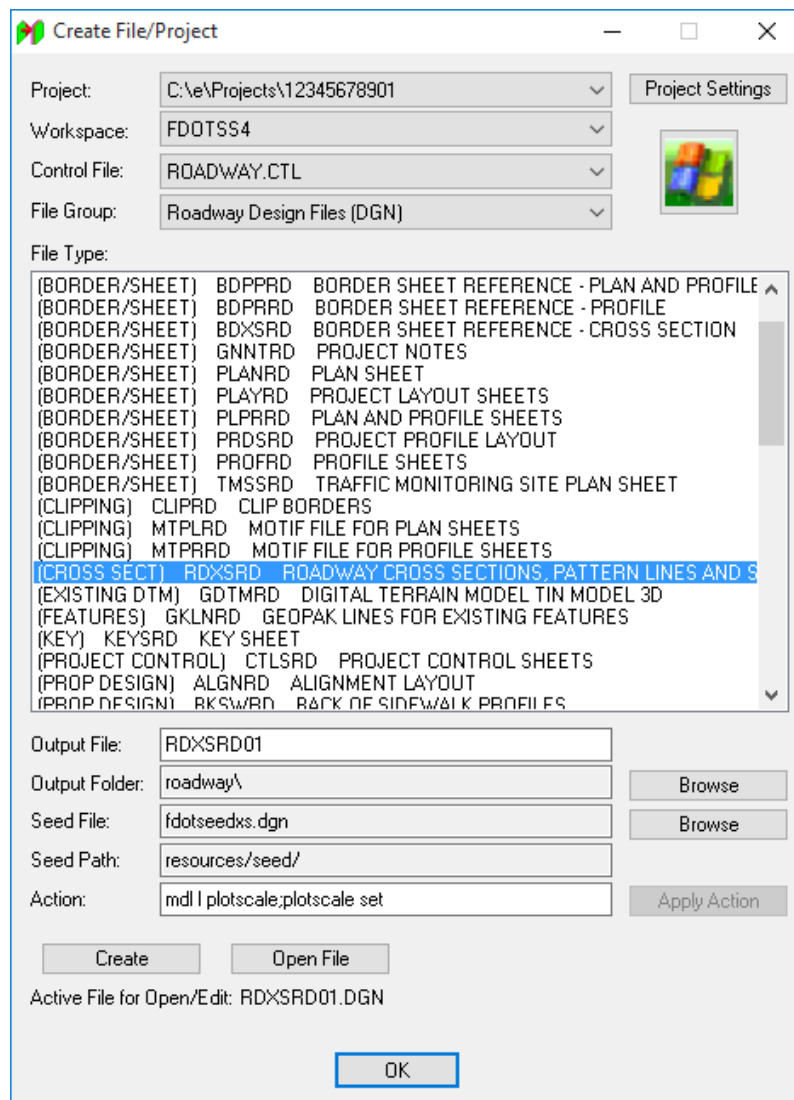
3. Click the **Create** button at the bottom of the Create Files dialog. Click **OK** on the Creates File confirmation dialog.

4. After the file is created, click the **Open File** button at the bottom of the Create File/Project dialog. MicroStation automatically closes the current design file and opens the newly created roadway cross section *rdxsrd.dgn* file. The Set/Update Plot Scale dialog displays.

5. From the Set/Update Plot Scale dialog, set *Scale* to **10** and *Units* to **English**, and then click **OK** to accept settings and continue.
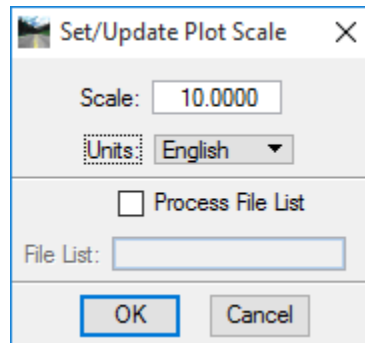
6. Click **OK** to close the Create File/Project dialog.

7. From the *Primary Tools* toolbox, select the **Models** icon to open the Models dialog.
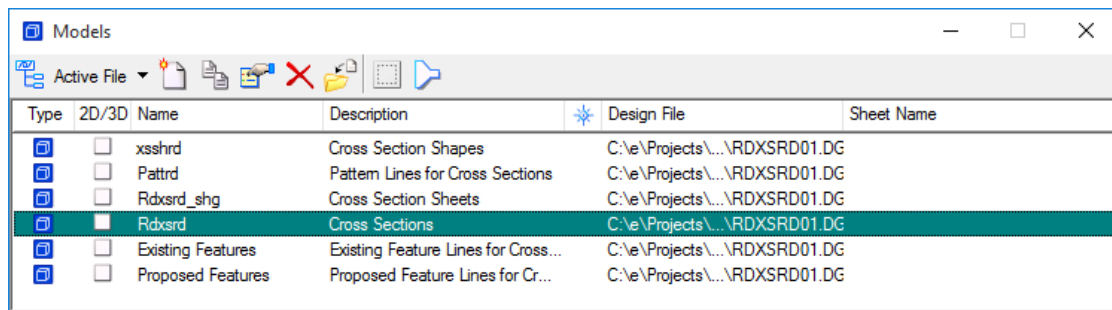
8. Review the models created in the roadway cross section file.

9. **Exit** MicroStation.

5 LEVELS

OBJECTIVES

This chapter focuses on leveling functionality in MicroStation, which is comprised of the following:

- Setting the Active Level
- Level Libraries
  - o By-Level Names
  - o By-Level Symbology
  - o Level Filters
- Level Manager
- Level Display

INTRODUCTION

A MicroStation design is made with basic building blocks called elements that are placed on a design plane within each model. Each element placed in a model is on a drawing *Level*. Levels are analogous to transparent overlays which can be combined in different ways to visualize different parts of a model. When creating a MicroStation design, the user can use many Levels to organize drawing information intelligently. Each Level can have its own *color, style,* and *weight symbology* along with a number of other attributes.

MicroStation has a limit of 4 billion levels that can be created and used in a drawing. Obviously, using such a high number of levels is impractical at best. FDOT has adopted a By-Level system utilizing FDOT Standard Level Libraries that define discipline specific *Levels* and associated *color, style,* and *weight symbology* for the production of all FDOT projects. These FDOT Standard Level Libraries are locked and cannot be modified by the user.

MicroStation provides for the management and display options for levels through Level Manager, Level Filters, and Level Display applications. FDOT has utilized these options in the setup of their CADD Standards as discussed in the following sections.

SETTING THE ACTIVE LEVEL

While a MicroStation design file has many levels available to the user, only one level is open for modification at any given time. This open level is referred to as the *Active Level*. All elements are created on the Active Level. It is important to change the Active Level to the desired level before placing elements. The level an element resides on may be changed later, but the best practice is to create it on the desired level to start with. The Active Level is the same in all views.

The following are ways to set the Active Level:

- The easiest way to set the Active Level is to click the *Active Level* combo box on the Attributes toolbox, Click the **Active Level** combo box to display available levels. Choose the desired level to be set as the Active Level.
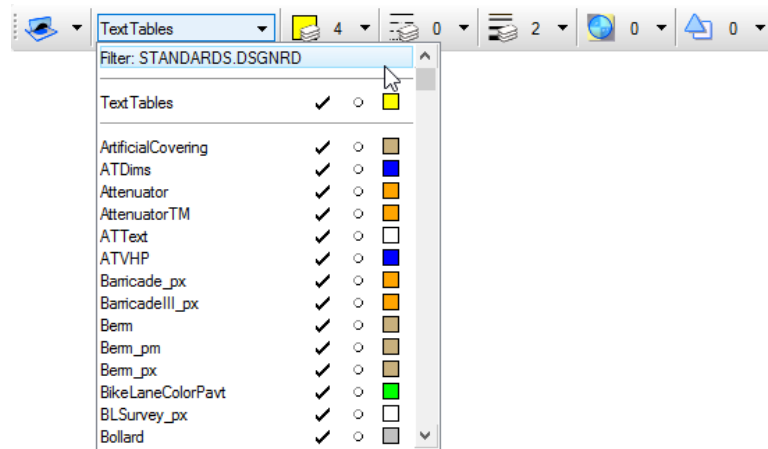
- The FDOT Menu contains a *Show Command List* icon that opens the Cadpilot UI TabList dialog. This dialog contains several tabs related to various disciplines with level lists relevant to each discipline. By selecting the logical level from one of the lists, the active MicroStation level is automatically switched to that selection.
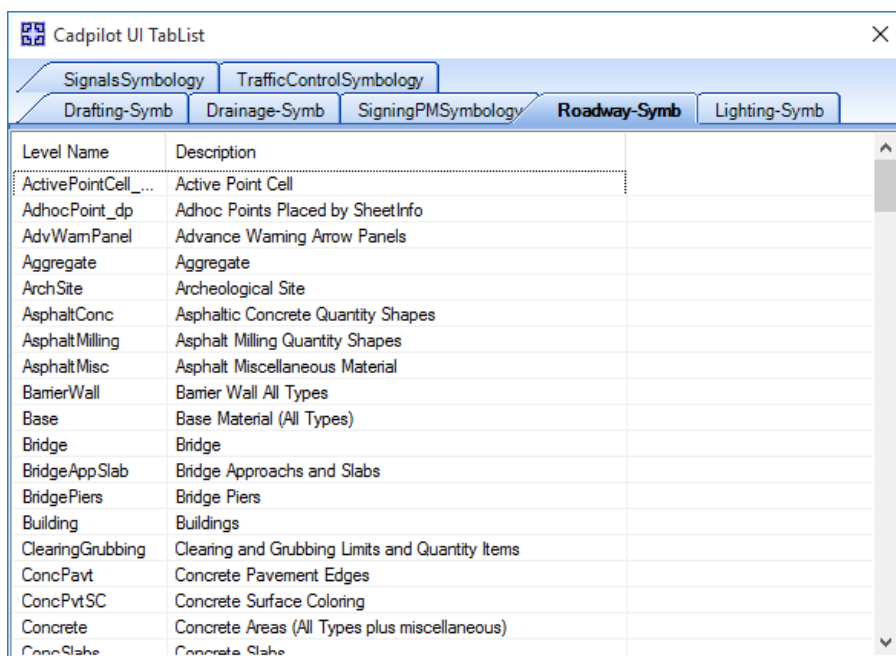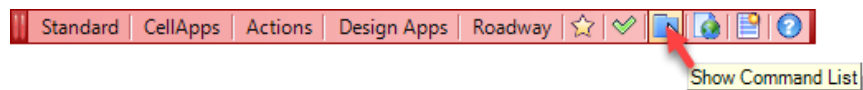
- From the Level Display dialog, the user can also set the Active Level. The user can click on the *Level Display* icon on the Primary Tools toolbox to open the Level Display dialog. The Active Level is highlighted in the Level list and can be changed by double-clicking on any other level <OR> by right-clicking on the desired level and select *Set Active* from the popup menu.
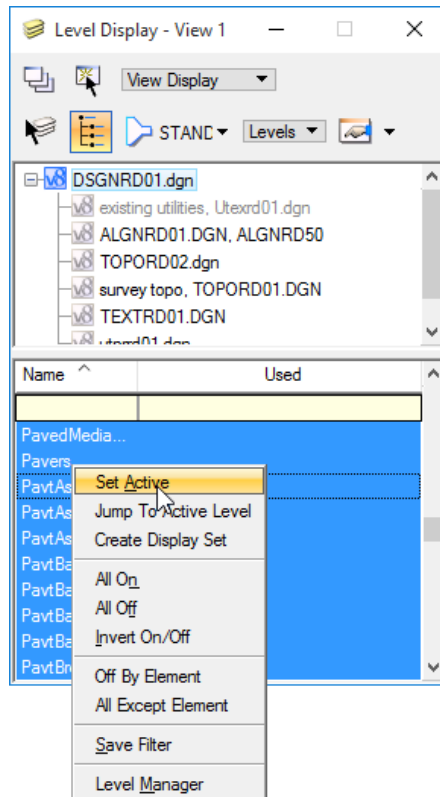
- From the Level Manager dialog, the user can also set the *Active Level*. The user can double-click the *Active Level* box in the MicroStation *Status Bar* to open Level Manager and then double-click on any level to set the *Active Level*.
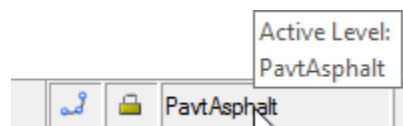
- From the Key-in dialog, enter ACTIVE LEVEL *<level – spec>* <OR> enter LV=*<level – spec>* Where *<level – spec>* specifies the level.

LEVEL LIBRARIES

A MicroStation *Level Library* refers to a component of a DGN Library that contains a level structure; that is, one or more level definitions. New level libraries can be created; and new and existing level libraries can be attached, detached, imported, and exported using the Level Manager dialog. A level definition does not technically attach from the Level Library to a design file *until it is used*.

As outlined in the CADD Production Criteria Handbook (CPCH), FDOT defines CADD level/symbology Standards using Design Libraries, Color Tables, Line Styles Resource files, Line Weights and Toolboxes. FDOT has created standard FDOT DGN Libraries (DGNLibs) within MicroStation to store FDOT Standards for: cells, levels, level filters, line styles, multi-line styles, text styles, dimension styles, element templates, menu customizations, customized tools, tool boxes, and tasks. Each FDOT DGNLib contains data that is shared throughout files and among users.

When a design file is opened in the FDOT Workspace in MicroStation, the corresponding FDOT Standards Level Libraries are automatically attached with preset FDOT Standard Filters activated. Only the valid levels from the attached libraries associated with the specific active design file are displayed for use.

The following is a listing of all standard FDOT Level DGNLibs:

DGNLib	DESCRIPTION
fdot_common_levels.dgnlib	FDOT Standard Common Levels
fdot_v8_levels.dgnlib	FDOT Standard Roadway Levels
rwlevels.dgnlib	FDOT Standard Right of Way Levels
StrLevels.dgnlib	FDOT Standard Structure Levels
survey_levels.dgnlib	FDOT Standard Survey Levels
V7_levels.dgnlib	FDOT Standard V7 Levels
countymappinglevels.dgnlib	FDOT Standard County Mapping Levels

BYLEVEL NAMES

Each FDOT Standard defined in a Design Library (DGNLib) is identified by a unique *Level Name*. When used from a DGNLib, it is copied into the active design file and is given the same name. This allows for comparison of the local resource to the FDOT DGNLib resource for compliancy to FDOT CADD Standards.

BYLEVEL SYMBOLOGY

The standard FDOT Level Libraries define the CADD Levels for each Discipline with the associated *ByLevel Color, ByLevel Line Style,* and *ByLevel Line Weight symbology*. Designers are to use these standard FDOT Levels to assign each element within FDOT CADD design files.

The FDOT Levels and symbology are grouped and translated into specific Rule Files which are associated to each valid Standard Filename of each Discipline for the purpose of performing the Quality Control check for FDOT Standard compliancy of design files. Complete specifications can be found in the CADD Production Criteria Handbook (CPCH).

When using MicroStation to create a FDOT drawing, the color, line style and line weight attributes should be set to "ByLevel". This setting allows the level to control these active attributes. This ensures that the levels comply with FDOT standards. FDOT levels are predefined and delivered in specific .dgnlib files. FDOT does not recommend overriding the ByLevel settings. To do so would result in non-standard design files that would not be acceptable in FDOT Project submittals.
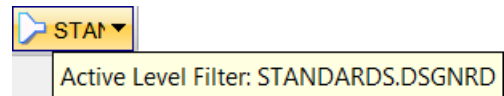
LEVEL FILTERS

MicroStation levels can be filtered at any time to make only specific levels within a Level Library available during the design process. FDOT utilizes the MicroStation level filters to combine level definitions within the FDOT Level Libraries into **Filter** groups.

FDOT created a Filter group called **Standards** that match the FDOT Standard Rule Files. Each **Standard** sub-filter is associated with an FDOT Standard filename. When a design file with an FDOT Standard filename is opened in MicroStation, only the FDOT Levels approved for use in that design file are displayed for selection.

These filter groups are not locked down and can be turned off at any time. The users also may create filters specific to their needs.

The FDOT Standard filter groups can be accessed:

- From the *Attributes* toolbox *Active Level Filter* icon. The user may hover the cursor over the icon to reveal the *Active Filter*. In this case, the Standards filter is DSGNRD:

    Active Level Filter: STANDARDS.DSGNRD

    Selecting the icon reveals the filter listing for selection, defaulting to the current *Active Level Filter* to the top of the display:

    DSGNRD
    DSGNSG
    DSGNSP
    DTMRD
    GEOTECH
    GSWKSP
    IRRGLD
    ITSSP
    KEYSHT
    MRARSP
    Numbered Levels
    PDXSRD
    PLANRD
    PLPRRD
    QTDSRD
    RDXSRD
    RDXSSP
    RWDTRD
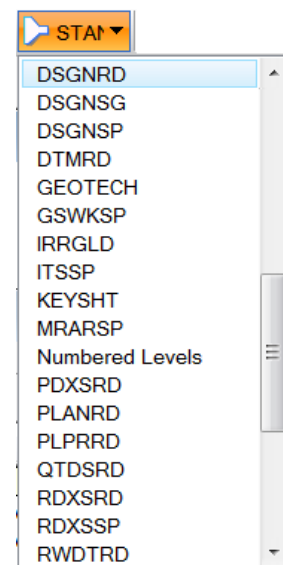
Note   If non-standard levels have been used within the active design file, the filters will omit these levels from display. The filters must be turned off to reveal all levels used within the active design file.

- From the Level Manager by selecting from the *List Filter* icon <OR> from the *Models, Levels, and Filters* list pane as discussed further in the next section.

    List Filter: STANDARDS.DSGNRD
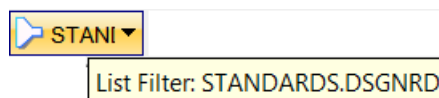
- From the Level Display by selecting from the *List Filter* icon.

    List Filter: STANDARDS.DSGNRD
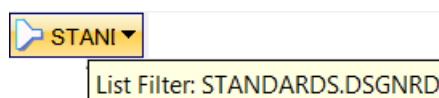
LEVEL MANAGER

MicroStation *Level Manager* is used to control level display and level symbology for the open DGN/DWG file and attached references. By default, the Level Manager includes a list of models, levels, and filters on the left, and a detail section of levels on the right. Multiple design files and/or reference attachments can be selected from the tree by holding the left mouse button down and dragging the cursor over the desired files. The levels for all the selected files display in the level list.

As previously stated, FDOT has established and delivers Standard CADD Symbology Levels within a set of DGNLIBs. FDOT's workspace disables the Level Manager's modification/creation functions of levels and does not allow users to create, delete, import or export levels to ensure consistency in products submitted for FDOT Projects. Every level needed for creating FDOT drawings should already exist, including miscellaneous levels for "scratch" drawing. If a new level is desired, a request must be submitted to be addressed through the CADD Technical Advisory Committee (TAC) of the appropriate discipline.

The Level Manager is accessed by selecting the Level Manager icon ( ) from the Primary tool bar

<OR> selecting the MicroStation menu option **Settings > Level > Manager**

<OR> by clicking in the Active Level field of the MicroStation Status Bar.
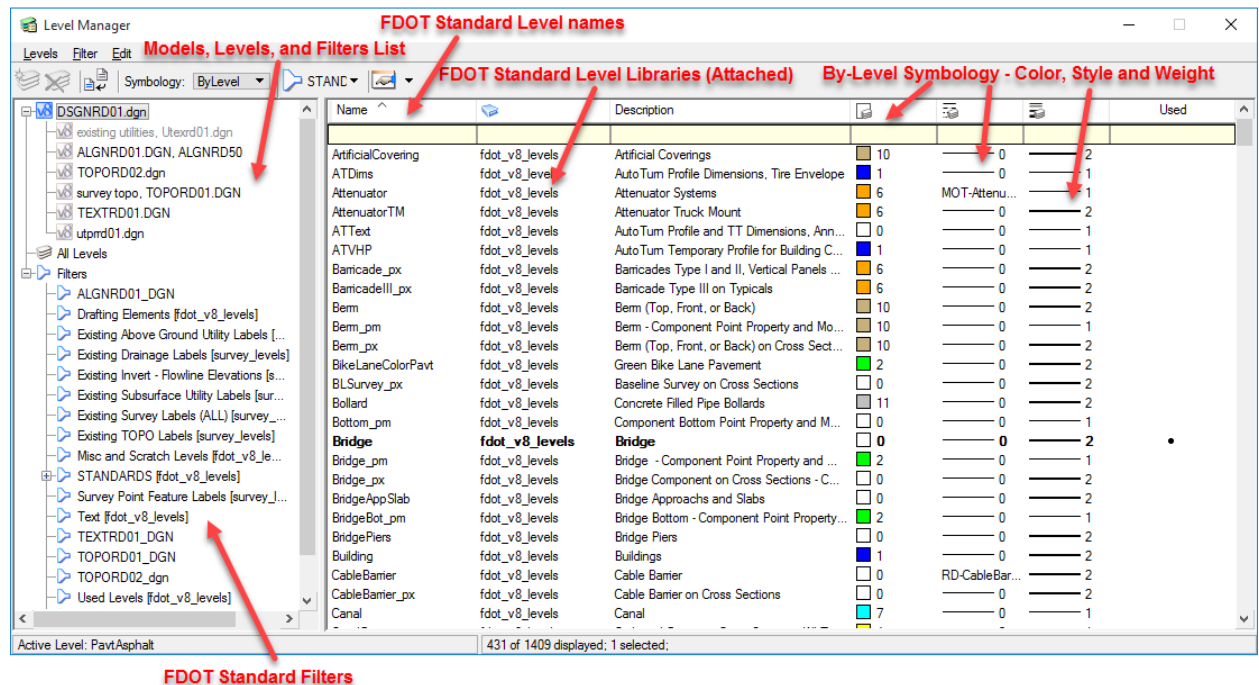
The Level Manager functions go beyond level editing, but provide much information to the user:

- **Model, Levels, and Filters List** - In the pane to the left of the dialog, the Level Manager displays the *Active Design File, Model* with all the references; the *All Levels* attached and used within the open design file and all *Filters* available for use by the user. FDOT has developed many filters from the various level libraries to match the FDOT CADD Standards and delivers them with the FDOT CADD Software. Upon opening of any FDOT Standard filename within the FDOT Workspace, an associated standard filter will be automatically applied, displaying only those levels valid for the active design file. The filter function can be disabled and / or utilized by the user to further customize other filters to best aide in the design process.
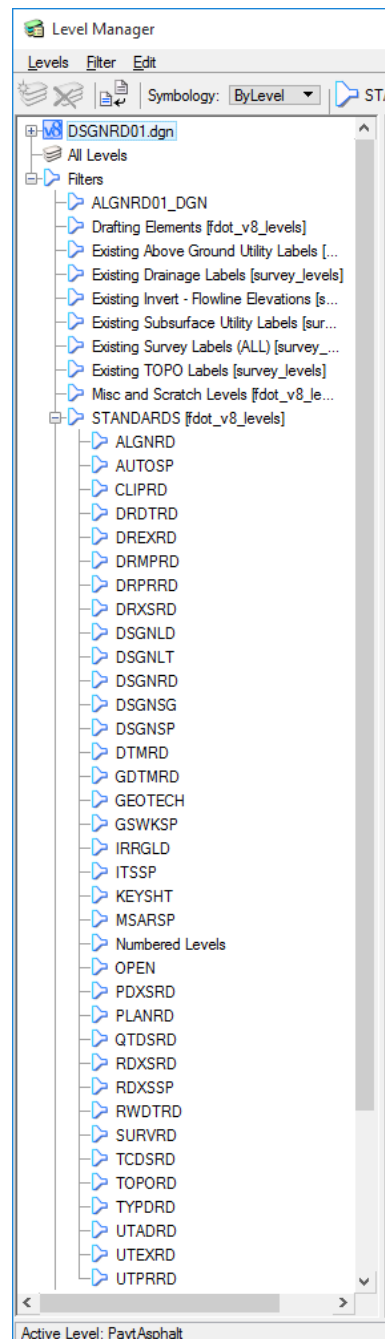
<table>
<tr><td>*Note*</td><td>The bracketed names refer to the FDOT Standard Level Libraries from which the filters are taken.</td></tr>
</table>

- **Levels List** – As previously stated, FDOT associates each FDOT Standard Filename with a Standard Filter. This listing displays the attributes and properties of individual levels allowed for the selected file or reference. The highlighted level indicates the *Active Level*. Bolded items represent levels that have been *used* within the design file. The listing can be sorted by double-clicking any of the titles. The columns in this listing can be customized by right-clicking on the title bar and checking or un-checking available options.
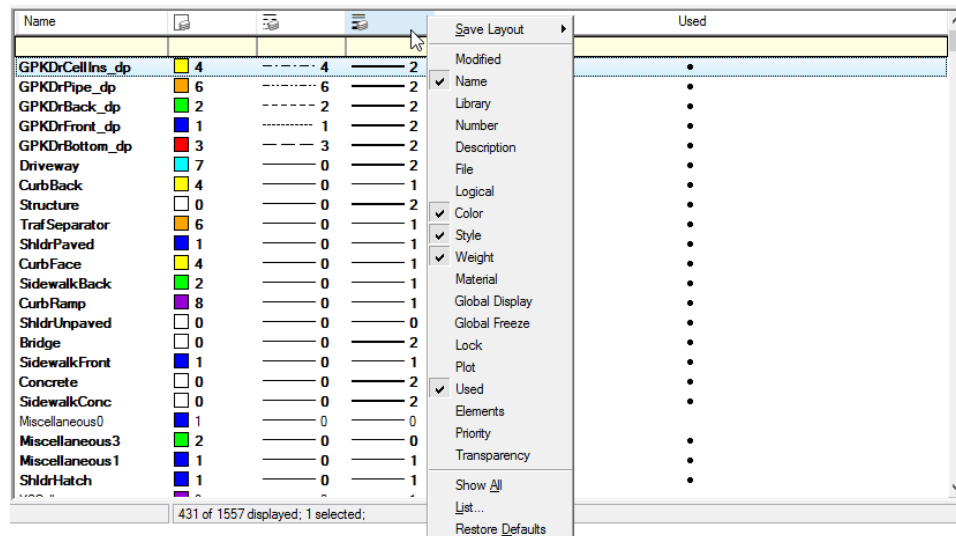
Note — To display all levels used, regardless of validity, the user must turn off all filters by selecting None from the dropdown.

- **Right-click popup** – Level Manager has a right-click option to allow the user to *Set Active* Level, *Jump To Active Level,* and view *Properties*.
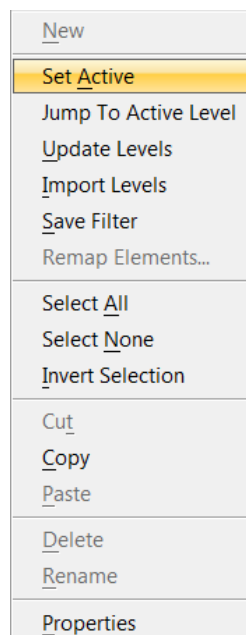
SYMBOLOGY OVERRIDES

Symbology Override is another control that can be assigned to a level, allowing the user to override the FDOT ByLevel symbology for that level. Once the Override is set in the level, the user can select the *Symbology* dropdown and set to *Overrides* to display those levels with an override setting.

Note   Overrides are NOT recommended by FDOT.

To set overrides in a level in the Level Manager, select the menu option **Levels > Properties** <OR> *right-click* on any level and select **Properties** from the popup menu. The Properties dialog opens and the user can review and/or change the override symbology in the bottom of the Level Properties dialog.
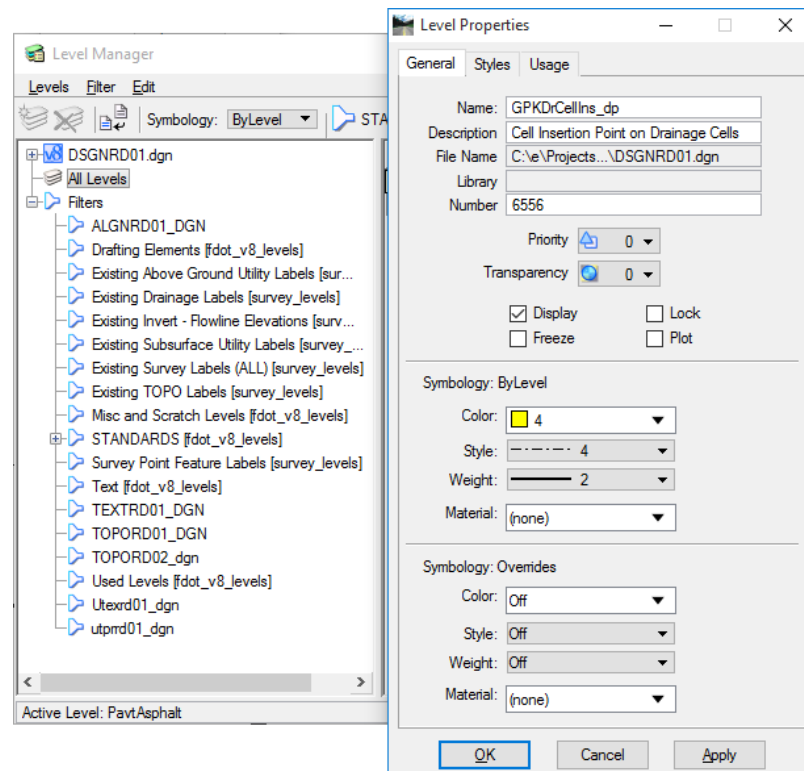
Multiple target files can be selected by the user to allow application of override symbology on specified levels existing in multiple files. For example, suppose several files have a level called Border and elements on that level appear as a different color in each file. The color symbology can be changed once so the color is changed in all of the selected files.

Note   From the View Attributes dialog, the Level Overrides must be activated for the override symbology to display.
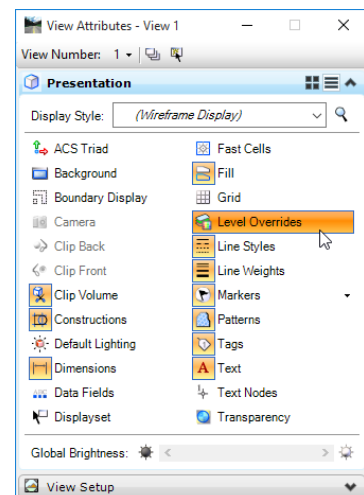
LEVEL DISPLAY

MicroStation gives the user a great advantage by allowing them the ability to control level display. The Level Display dialog provides for this action to turn on and turn off levels in a model and to apply filters created in the Level Manager dialog.
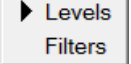
This dialog can be accessed through the MicroStation menu option: **Settings > Level > Display**

<OR> by selecting the hotkeys **[Ctrl] + [E]**

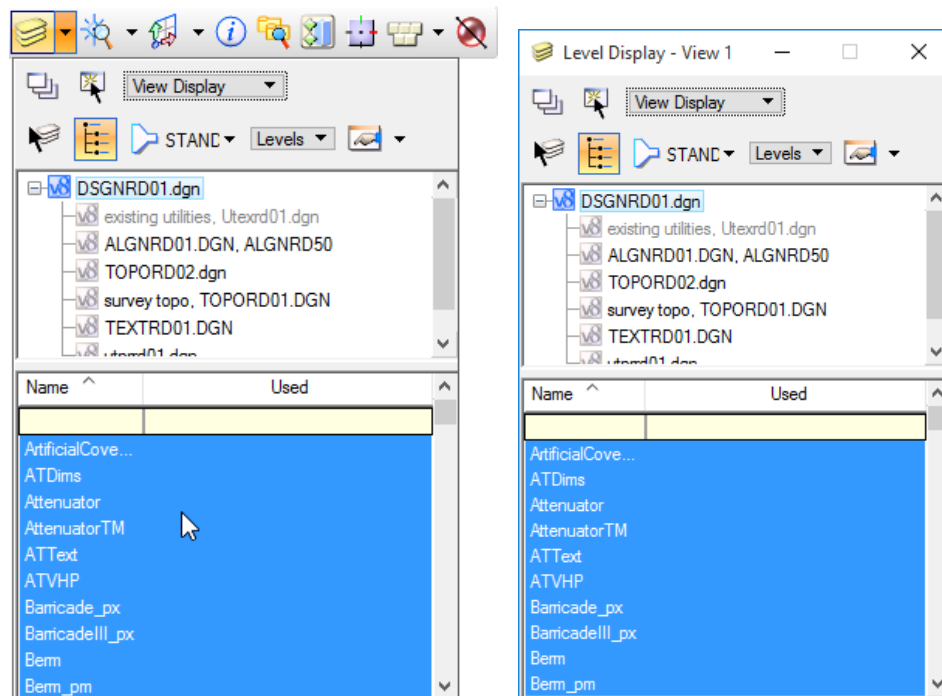<OR> From the *Primary* toolbox, the user can select the *Level Display* icon to open a floating dialog. Note the down arrow next to the icon allows the Level Display to open in popup mode and automatically close when the cursor is moved off of the dialog.

- **Apply to Open Views** – If on, level display settings adjustments will apply to open views.

-   **Apply to Selected Views** – If on, level display settings adjustments will apply to the selected view.

- **Level Display Setting (Optional)** – Sets the level display for the dialog. Turning these options ON or OFF is the same as thawing or freezing levels, with one difference. These settings determine whether any cells or references placed on that level will or will not be displayed.

  - ✓ *View Display* – Changes in the level display affect the chosen view in the active model.

  - ✓ *Global Display* – Changes in the level display affect all views in all models in the open file.

  - ✓ *Global Freeze* – (DGN files only) Indicates whether the level is frozen. If frozen, elements on the level are not displayed and cannot be printed.

- **Change Level** – Opens the Change Level tool, used to display or lock a level.

- **Show Target Tree** – Changes the display of the Target Tree, used to set the target mode for level display settings changes or sets the model as a target.

- **List Filter** – If the **Show Level Name or Filters** option menu is set to *Level,* this option will allow the user to set a Filter, named or defined on-the-fly, applied to level display.

- **Show Level Name or Filters** – Determines the contents of the List box.
  - ✓ *Filters* – Lists level filters defined in the open file (and in reference attachments, if they are selected in the target tree).
  - ✓ *Levels* – Lists the levels in the open file (and in reference attachments, if they are selected in the target tree).

- **Dialog Properties** – Pops up a Level Display dialog, used to set the properties of the Level Display dialog.

- **Target Tree** – Used to select the target model for level display settings changes. This tree control enables selection of the active model, another model in the open DGN file, or an attached reference in the open file or another file, as the target.

  Right-clicking on the active file or the reference files in the target tree area opens popup menus with the following options:
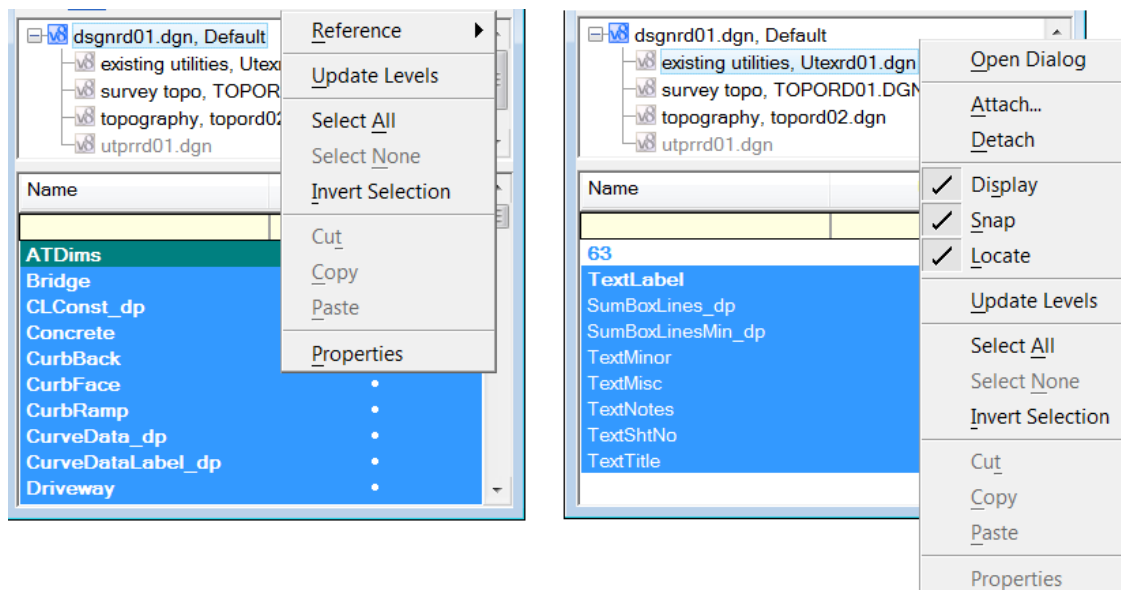
Levels (List box) – Lists the Levels (or filters if *Show Level Names or Filters* is set to **Filters**) in the target model. This listing has the same display, sorting, and function capabilities as the Level Manager.

Right-clicking on the title bar opens a popup menu to customize the list display columns with the following options:
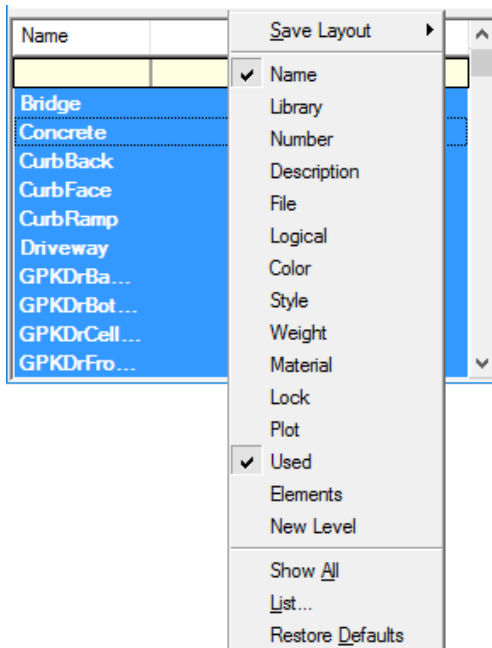
Right-clicking within the listing opens popup menu with the following function options:

WORKING WITH LEVELS

CHANGE ATTRIBUTES CLASSIC

One set of tools that works with levels is the Change Attributes toolbox. The tools contained in this toolbox can be used to easily move elements from one level to another. They can also be used to match the current attribute settings to an existing element.

The Change Attributes toolbox is accessed by selecting the icon ( ) from the Main Task <OR> by drilling down the Task Navigation tool <OR> by checking on the toolbox through the MicroStation menu option **Tools > Toolboxes**.
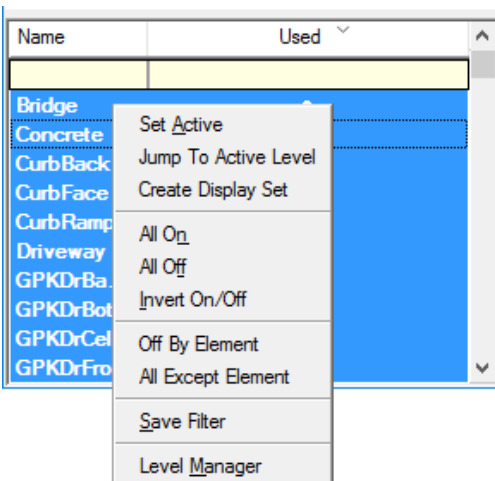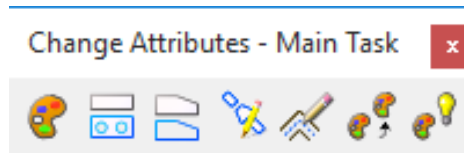

	Change Element Attributes	Used to change an element(s) level, color, line style, line weight, or class. This tool may also be used to match one element's attribute to another's.
	Change to Active Area	Used to change the area attribute of a closed element(s) (shapes, ellipses, complex shapes, or B-spline curves) to the Active Area (Solid or Hole).
	Change Element Fill Type	Used to change a closed element fill type attributes (shape, ellipse, complex shape, or B-spline curve) to the Active Fill Type.
	Modify Line Style Attributes	Used to interactively modify the line style attributes of an element with a custom line style. The tool setting *Method* determines the type of modification.
	Change Multi-line	Used to change a multi-line's attributes to the active multi-line definition.
	Match Element Attributes	Used to change the active element attribute settings so they match the attributes of an existing element.
	SmartMatch	Used to change all active element attribute settings, including those specific to particular element types, so they match the attributes of an existing element. When a cell is matched, the active scale factors and Active Cell attributes are also matched.

CHANGE ELEMENT ATTRIBUTES

From the *Change Element Attributes* tool setting dialog, the user can control what properties of an elements' symbology are changed. Of all of the tools in the Change Attributes toolbox, Change Element Attributes is the most commonly used.

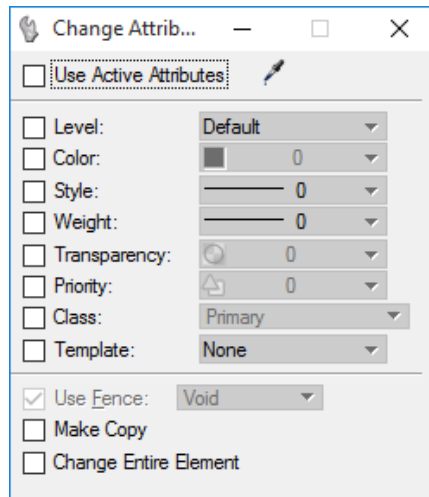|	
Use Active Attributes	If on, active attributes are affected when the user changes or matches attribute settings. If off (default), the active attributes settings are not affected when the user changes or matches attribute settings.
Eyedropper icon (Match Element Attributes)	Used to match the element attributes of the element selected, and then use these settings to change the attributes of the next element(s) selected. If *Use Active Attributes* is *on* when clicking the eyedropper, the active attributes will change to match the element selected. If *Use Active Attributes* is *off* when clicking the eyedropper, only the settings in the tool settings window change to match the element selected. The active attributes do not change.
Level	If on, level is included when changing or matching settings.
Color	If on, color is included when changing or matching settings.
Style	If on, line style (and any active line style modifiers) is included when changing or matching settings.
Weight	If on, line weight is included when changing or matching settings.
Transparency	If on, element transparency is included when changing or matching settings.
Priority	(2D only) If on, element priority, is included when changing or matching settings.
Class	If on, element class is included when changing or matching settings.
Use Fence	If on, the tool will modify the element(s) based on the fence and fence setting used. Note that this option is grayed out unless there is an active fence placed in the design file.
Make a Copy	If on, the element(s) are copied and the attributes of copy(s) are changed, but the attributes of the original are not changed. Turn this setting on in conjunction with the Level setting to copy an element from one level to another or from a reference to a master file level.
Change Entire Element	If on, the attributes for the entire element are changed.

       FDOT MicroStation Essentials - Part I

Exercise 5.1 *Level Manager*

➢ *Reviewing Level Manager*

1. Open the MicroStation file c:\e\projects\12345678901\roadway\dsgnrd01.dgn.

2. From the Primary Tools toolbox, select the Level Manager ![Level Manager icon] icon.
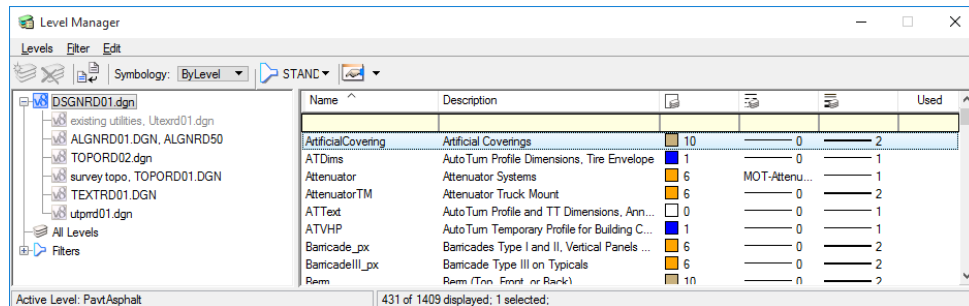
3. The FDOT CADD Software delivery includes an FDOT Workspace with a preset Level Library that is locked, disallowing the user from creating new levels. Also included are preset MicroStation Level Filters that are automatically associated to all FDOT Standard Files.

4. From Level Manager, move the cursor to hover over the Filter List ![STANI icon] icon. Note the Standards filter, DSGNRD that has been automatically associated and displayed.

5. Select the down arrow to open the popup listing and then and scroll to the top to select (none). The Level list pane updates with all available levels from the Level Libraries attached to the design file.
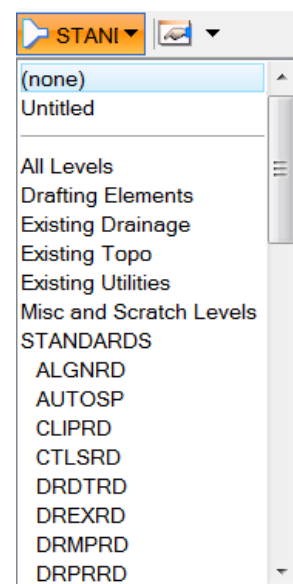
> *Note*   Not all levels are embedded into the design file, only the levels used displayed in bold in the Level Manager.

6. From the Level List, click on the column titled Used to sort all the Levels used within the design file to the top of the list. The title may have to be toggled twice as the list can be sort ascending or descending. Select other column title headers and note how each sort ascending or descending.
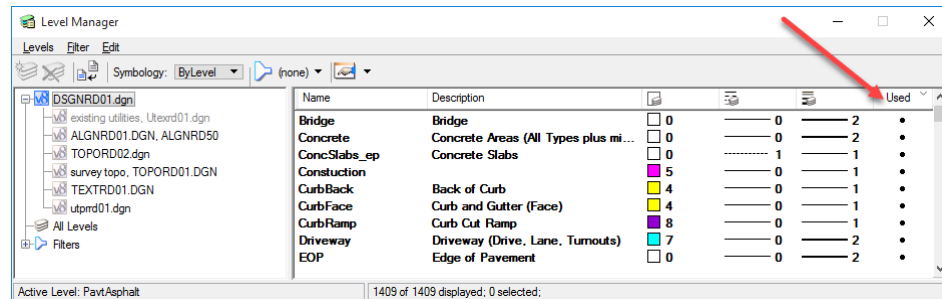
7. Right-click the *Level List* and select **Jump to Active Level** from the *popup* menu. The *Level List* moves to and highlights the current Active Level as displayed in the Attributes toolbar.
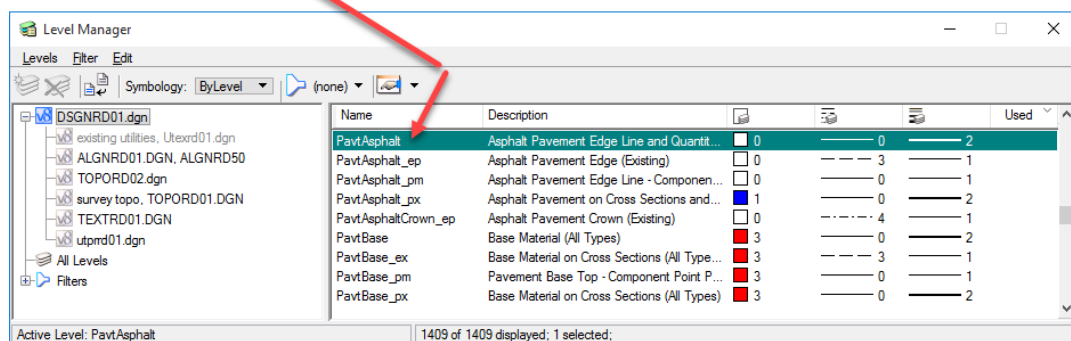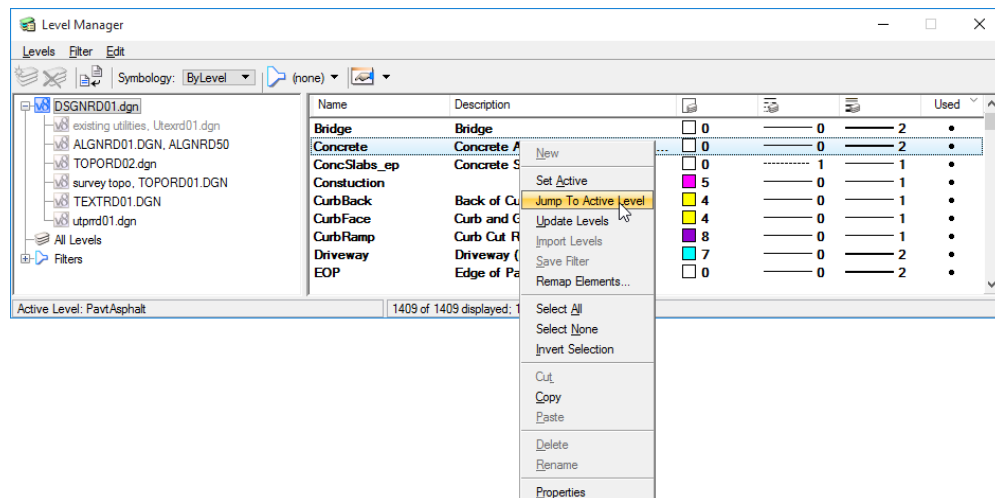
8. From *Model, Levels, and Filters List* pane to the left of the window, expand the **Filter** sections by clicking the ⊞ icons. Note the listing of preset *FDOT Filters* and what [Level Library] they are drawn from.
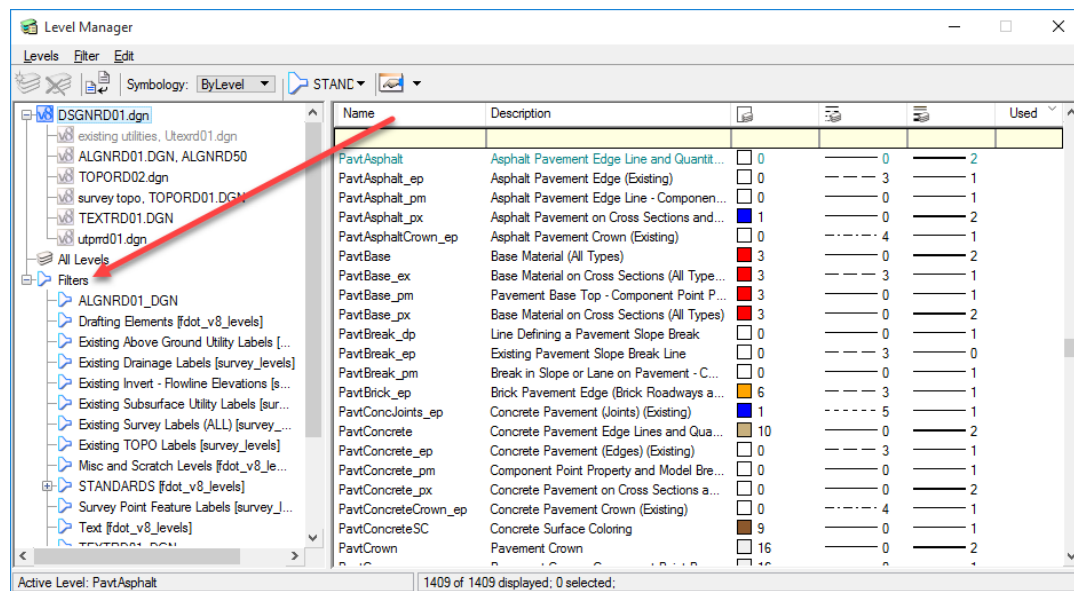
9. Expand the *STANDARDS [fdot_v8_levels]* parent filter by clicking the ⊞ icon. All the filters displayed match directly to the FDOT Standard Rules. Select the **DSGNRD** to display the FDOT Standard Levels allowable for use in the current active design file, *dsgnrd01.dgn*.
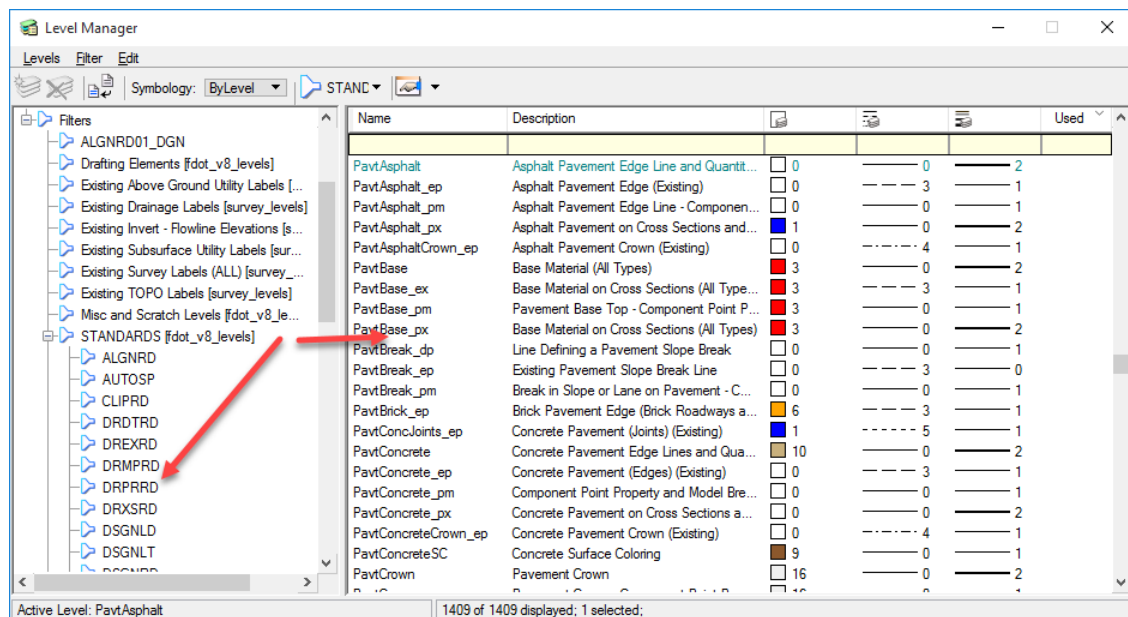
10. Close the Level Manager.

Exercise 5.2 *Setting the Active Level*

> ### ➢ *Active Level*

1. Continuing with the *dsgnrd01.dgn* file, zoom into the **Precious Pet Tax Collectors** office located near **Station 273+00**.

2. From the Attributes toolbar located at the top of the application window, set the *Active Level* to **Fence**.
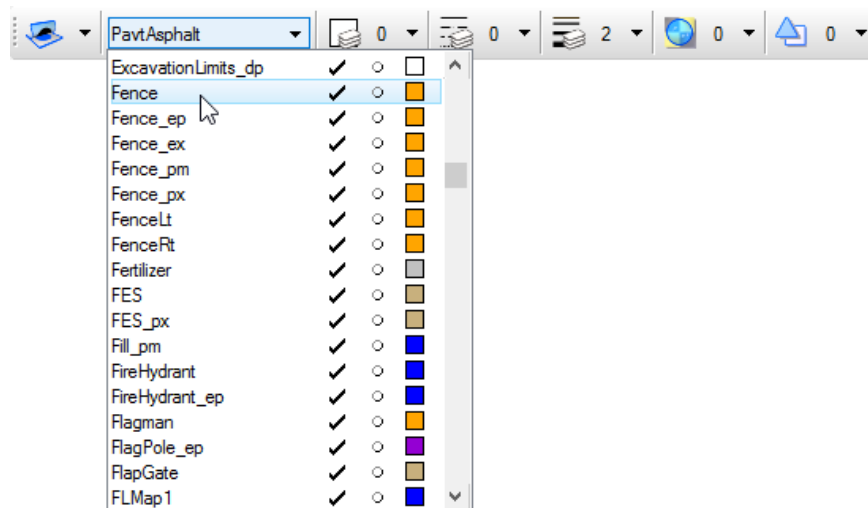
3. From the Task Dialog under the Drawing task, the **Place Line** tool. The Place Line Tool Settings box displays. For this exercise make no entries.
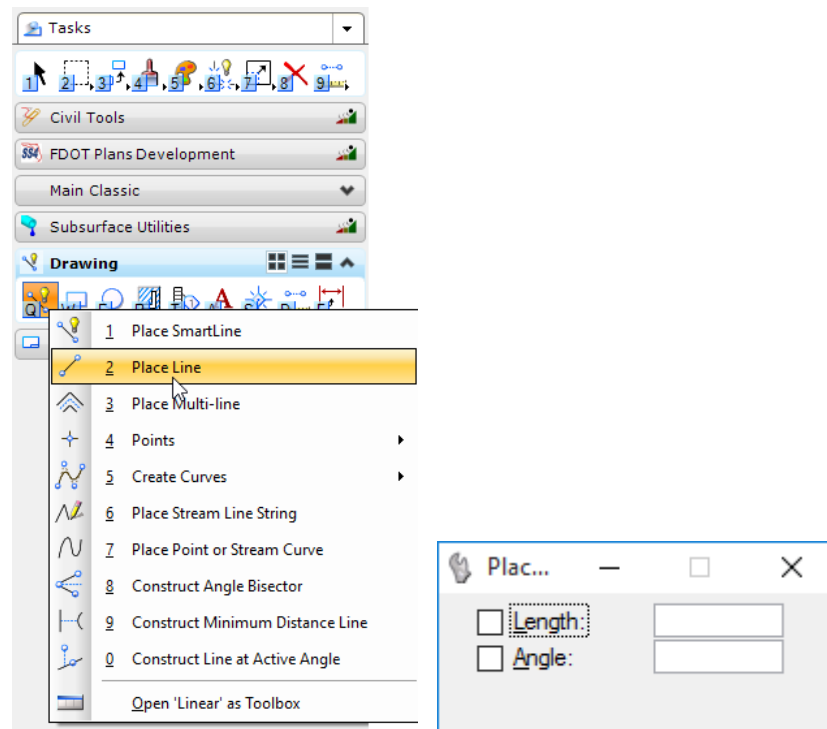
4. Draw two Fence lines at the front of the Tax Collectors office by placing **Data Points** at *the start, corner* and *end* locations as shown below. Right click to reset and end process after each line.
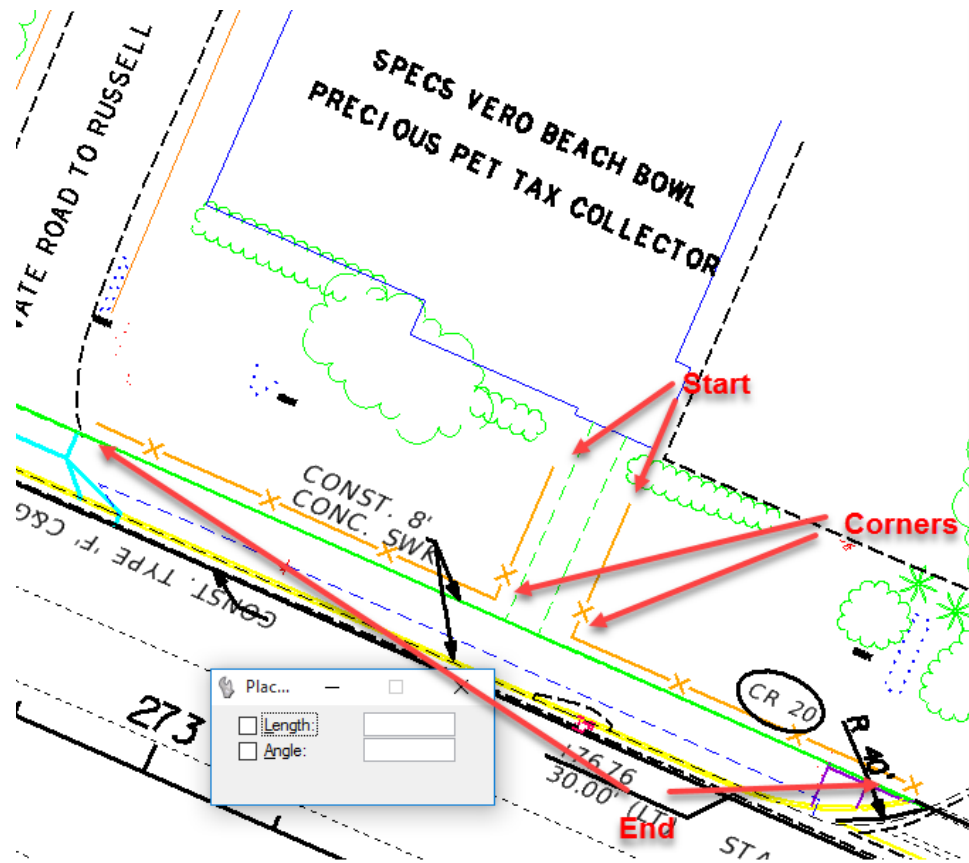
➢ *Level Override*

1. Continuing in the same location, from the *Attributes* toolbar located at the top of the applications window, select the Level Manager icon.

2. From the Level Manager, scroll down *the Level List* box and highlight the *Level* **Fence**.
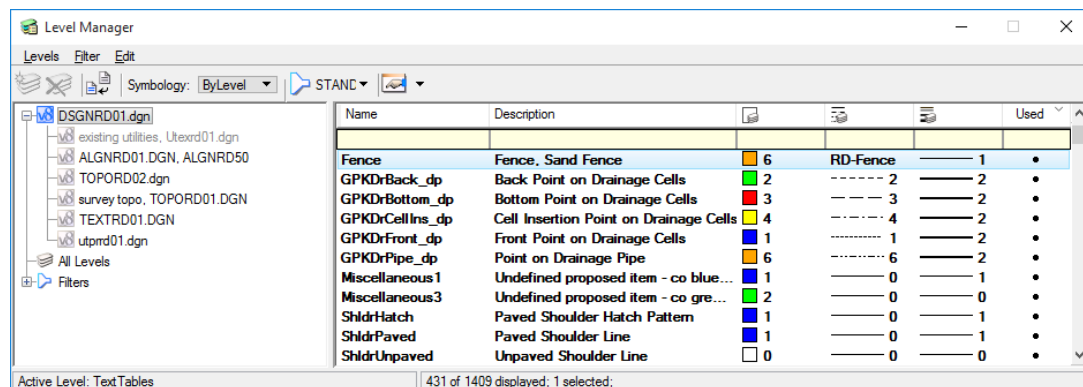
3. From the *Level List*, right-click and select **Properties** from the *popup* menu. The Level Properties dialog opens displaying all the attributes of the highlighted level. Review each of the tabs.
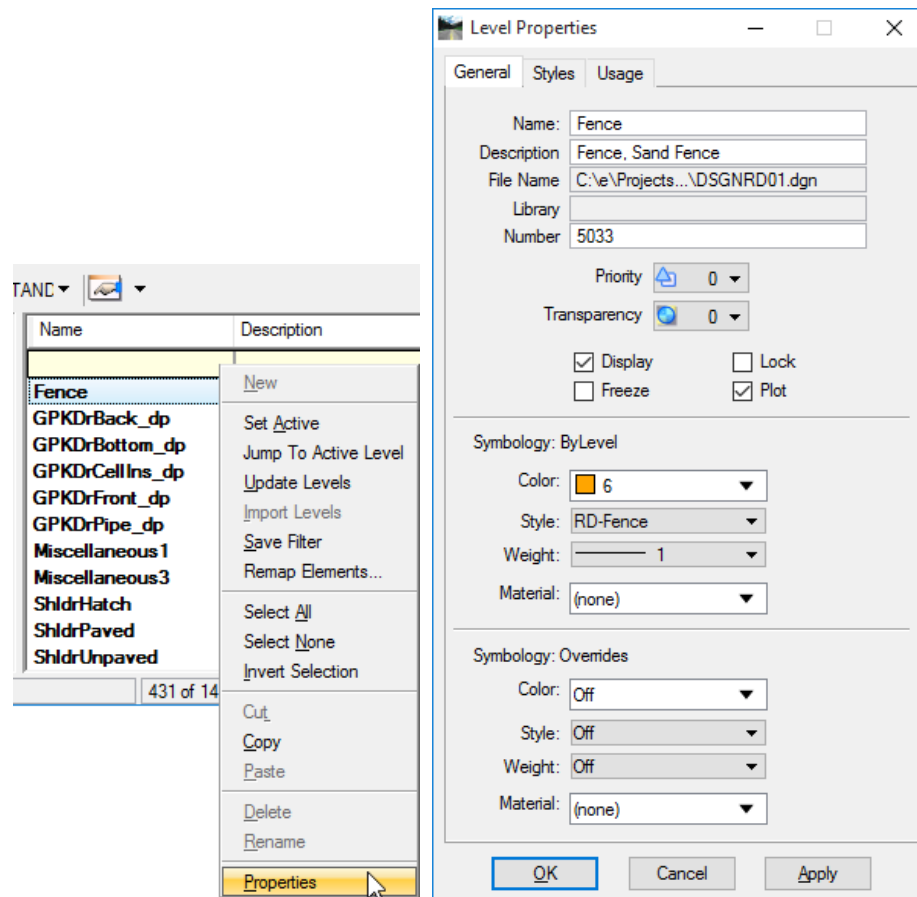
4. From the Level Properties dialog, under the **Symbology: Overrides** section change the *Attributes* by clicking the down arrows and then selecting as shown below.
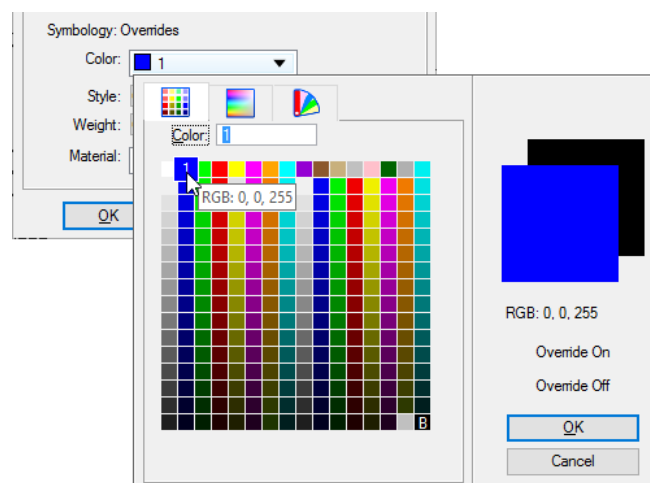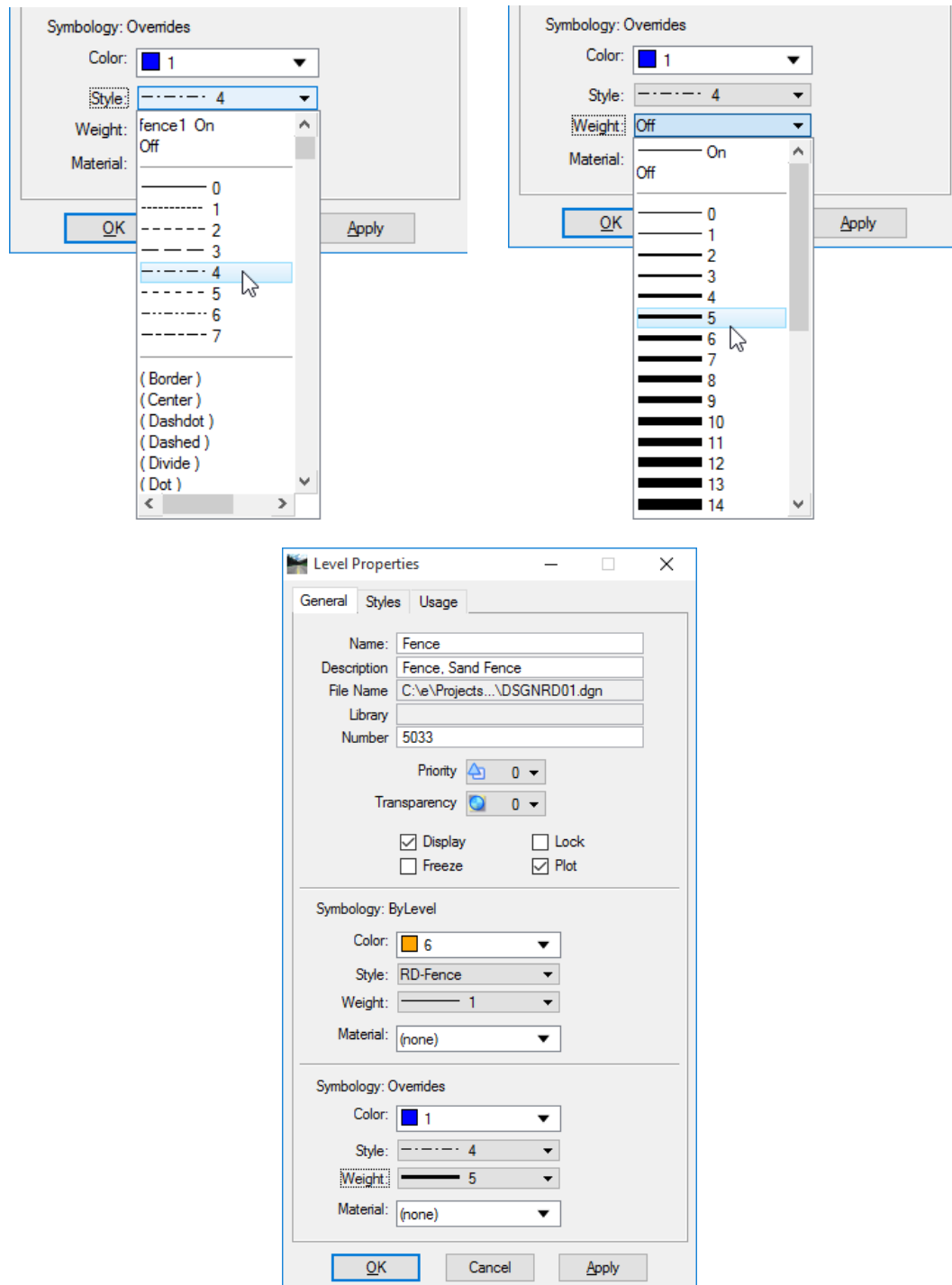
Note   Settings overrides in this dialog would change all the attributes of all the elements listed under the Usage tab within the active design file and cause them to become Non-Standard (non-compliant) to the FDOT Standards.

5.   Clicking **Apply** saves the Override changes with the dialog remaining open. Click **OK** and the changes save and the dialog disappears.

6.   Close the Level Manager.

7. Review the graphics and the affect the override symbology has.
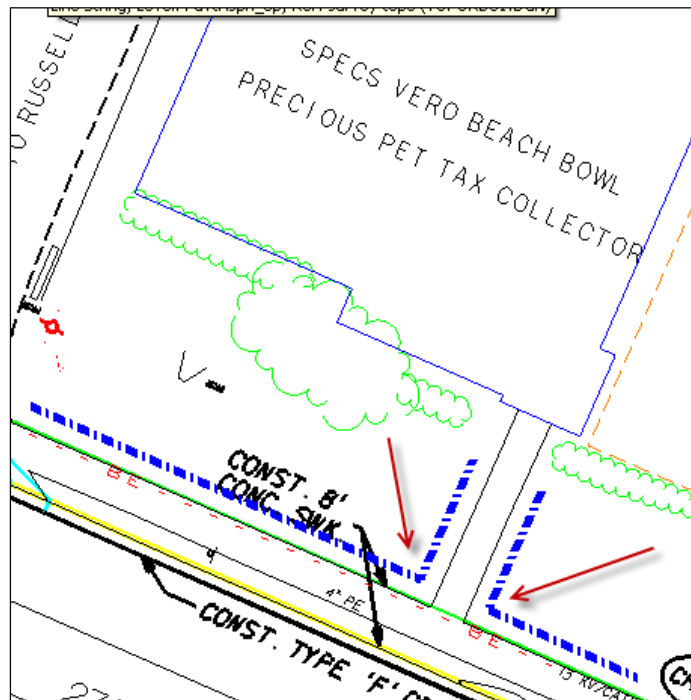
8. From the *View Controls*, select the View Attributes ▭▾ icon.

9. From the View Attributes dialog, toggle **OFF** (un-highlight) the option for **Level Overrides** and notice the affect on the symbology of the *Fence Line* within the drawing.
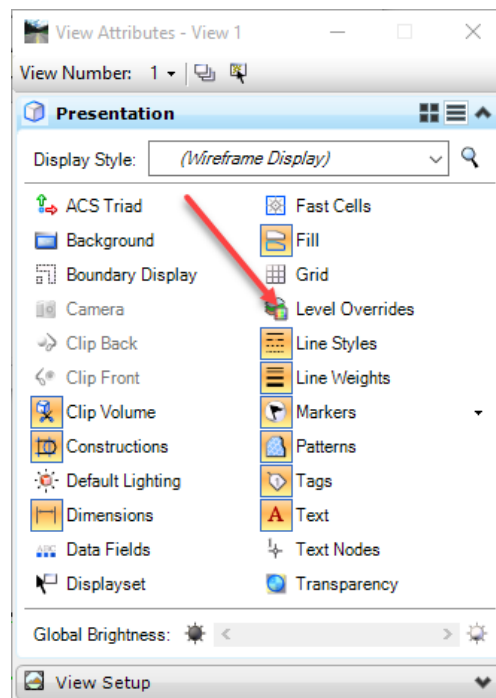
10. Close the View Attributes dialog and exit MicroStation.

6 **BASIC DRAWING TOOLS**

OBJECTIVES

This chapter reviews the understanding of the basic drawing tools provided in MicroStation, including:

- Placing Elements
- Tool Settings
- Basic Workflow
- Linear Task
- Polygons Task
- Arcs Task
- Ellipses Task
- Undo and Redo

INTRODUCTION

MicroStation has dozens of drawing tools (or tools, for short). These tools are grouped for convenient selection in toolboxes and are referenced in tasks. Tools referenced in a single task are used to perform a particular task, functionally speaking. For example, the Drawing Composition task references tools used to compose drawings. Only one task is active at any given time. The tools in the active task are grouped in a special dialog — the Tasks dialog.

Each open toolbox is either floating in its own window or docked to an edge of the application window. The arrangement of tools in a floating toolbox can be changed by resizing its window. Tools are represented in toolboxes by icons. For simplicity, the term "tool" is used to refer both to a tool and its icon.

Within MicroStation, one tool or another is selected, or active, at all times. The default selected tool is *Element Selection*, which is used to select elements for manipulation or modification. When a design file is opened in MicroStation, the Element Selection tool is automatically selected or active.
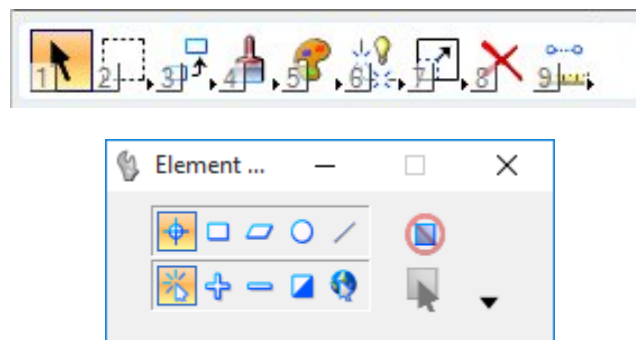
As tools are selected and used within a MicroStation session, the last tool selected by the user becomes the default tool.

FDOT has developed and delivers within the FDOT Workspace within MicroStation, many discipline-specific Tools, Toolboxes, Tasks and Workflows to assist the user in designing and producing projects to meet FDOT CADD Standards. These have been incorporated into the Tasks Dialog as shown below.
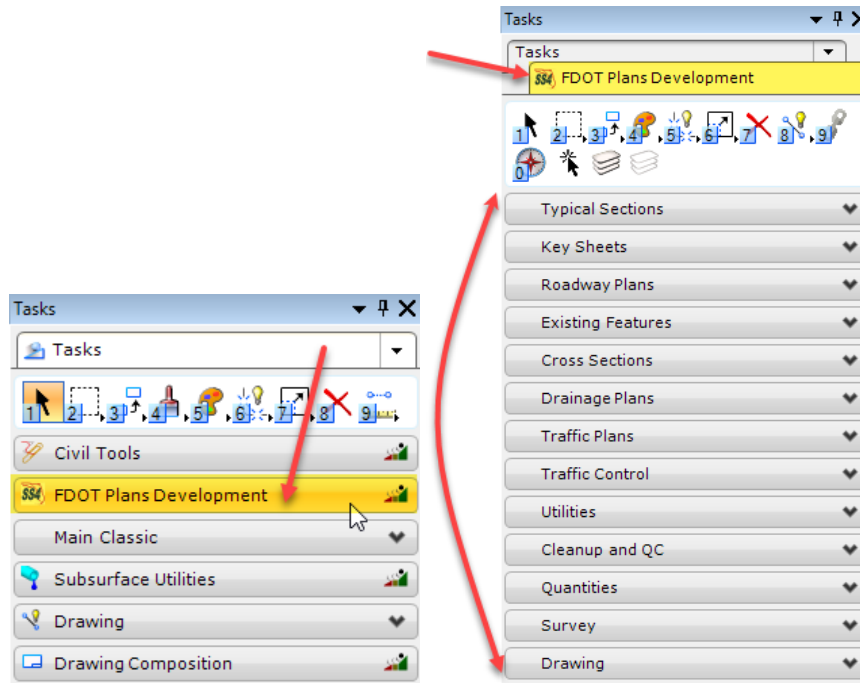
PLACING ELEMENTS

The objects drawn, or placed, in a model, lines, polygons, circles, text, etc., are called *elements*. Each element has several attributes that define the element:

- Level

- Color

- Line weight

- Line style

- Fill color (for closed elements)

- Class

- Transparency

- Priority (2D only, for display)

FDOT defines the CADD level/symbology Standards for Level, Color, Line Weight and Line Style and has incorporated these standards in the FDOT tools delivered within the FDOT Workspace. The users need only to utilize these tools to create elements set to the required level/symbology to pass compliancy with the FDOT CADD Standards.

In MicroStation, the left mouse button is mapped to a logical button called the *Data* button, which sends a type of graphic input called a *data point* when pressed. The right-hand mouse button is mapped to the *Reset* button, which is used for resetting and for accessing Reset pop-up menu and View Control pop-up menu. The pressing of both right and left hand mouse buttons at the same time, called a button "chord", invokes the *Tentative* button, which is used to enter tentative snap points. If a three button mouse is available, it can have the center button mapped as the tentative button.

When selecting an element within a design file <OR> or when placing the cursor into the design plane of a design file to create or manipulate an element, an element is being *positioned* to a specific location. These positioning locations are called *data points*. A data point is a graphic input that, depending on the context:

- Designates a point in a design (for element placement or selection, for example).

- Designates the view in which it is entered (for fitting or updating, for example).

- Accepts an operation (deleting an element, for example), rather than rejecting it.

While dragging to enter data points is fast and easy, the preferred technique is clicking to enter each data point for more precise accuracy.

TOOL SETTINGS

MicroStation has several aids to help to use tools. Notice a floating dialog whose title and contents change each time a tool or view control is selected. This dialog is the *Tool Settings* window. Almost every MicroStation tool has settings that affect how the tool works and can be controlled in this Tool Settings window. If closed, the window opens automatically when a tool with settings is selected. The status bar along the bottom border of the application window displays prompts and messages about the selected tool.
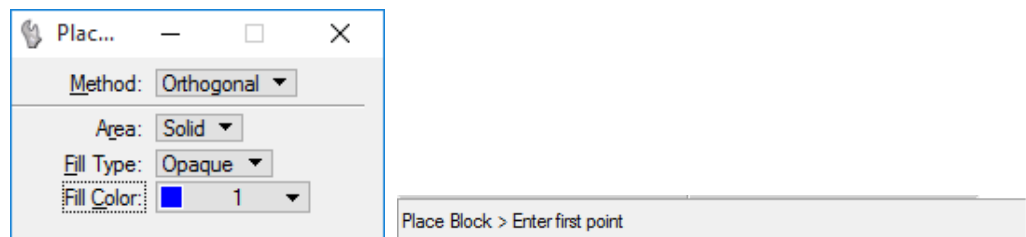
Rather than forcing the adjustment of a tool's settings each time the tool is selected, tool settings remain in effect until changing them. This makes the product more efficient to use, but also means the active tool's settings must be kept in mind or displayed on the screen.

Ordinarily upon selecting a tool, the settings are adjusted before entering any data points. However, some tool settings are not applicable until data points are entered.

For example, while using Place SmartLine, after entering three or more data points and then move the pointer near the location of the first data point, a check box labeled Closed Element will appear in the tool settings window, enabling to the placement of a polygon.

Tip   *If using the default function key menu, press <F10> to open the Tool Settings window. If already open, pressing <F10> sends the input focus to the Tool Settings window.*

BASIC WORKFLOW

MicroStation has a basic workflow as a simple foundation to follow and remember when using drawing tools:

1. **Set the Active Element Attributes**. The recommended practice is to set the active element attributes first, even though, these can be changed after placement. It is advisable to always be aware of the active element attribute settings when placing elements.

   FDOT CADD Standards are based on pre-determined By-Level settings. For FDOT projects, the Level attribute need only to be set and the Color, Line Style and Line Weight will automatically set to the By-Level settings.

Note   Although not recommended, these settings can be overridden, if necessary, but will require an approval from the appropriate authorities of the FDOT project.
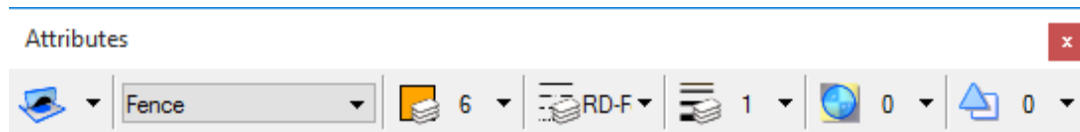
This step can be skipped if not placing an element or using a tool that does not rely on element attribute selection.

2. **Select the Tool.** All Tools and Toolboxes in which the tools reside are accessible through the MicroStation Tools menu or Task Dialog or the FDOT Menu. Throughout the learning process, experiment with different tools and utilize the *Undo* and *Redo* functions as needed.

3. **Adjust the Tool Settings.** When selecting different tools, review each of the Tool Settings functions as the window changes. The MicroStation help document contains descriptions of all tool settings. The specific help can be accessed by clicking in the tool setting window and then press <F1>.
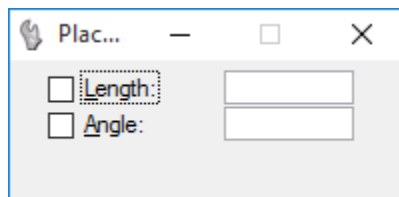
Tip   *The default Tool Settings is Element Selection, if no other tool is selected. Pressing <F10> sends the input focus to the Tool Settings window. The Tool Setting of the last selected tool will always display.*

4. **Follow the Status Bar Prompts.** In the beginning, remember to follow the prompts in the MicroStation Status Bar located in the lower left corner of the application window. These tell how to proceed from step to step.

LINEAR TASK

Lines are the most basic element in any design file. No matter what discipline is involved, linear elements play a large role in the basic geometry. The Linear Elements toolbox contains many tools that place linear elements into a design file.
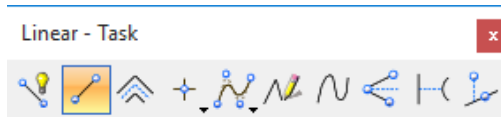
	Place SmartLine	Used to place a line, line string, shape, arc, or circle or a combination thereof as a complex element.
	Place Line	Used to place or construct a line.
	Place MultiLine	Used to place a multi-line, which can be made of up to 16 independently defined lines.
	Points Task	Used to place the Active Point.
	Create Curves Task	Used to draw curves and extract iso-parametric lines from a solid or B-spline surface.
	Place Stream Line String	Used to place a stream line string; primarily for tracing images when digitizing.
	Place Point or Stream Curve	Used to place a curve using the points or stream method.
	Construct Angle Bisector	Used to construct a line that bisects an angle defined by three data points.
	Construct Minimum Distance Line	Used to construct a line between two elements at their closest points.
	Construct Line at Active Angle	Used to construct a line that intersects a line segment (line or segment of a line string or shape) at a defined Active Angle.

Exercise 6.1 *Placing Linear Elements*

➤ *Place Line*

1. Open the MicroStation File c:\e\projects\12345678901\roadway*dsgnrd01.dgn*.

2. Zoom into the location of **Station 206+00 to 208+00** at the beginning of the project. This is the location where a sidewalk will be placed along the left side of the roadway.

3. **Set the Active Element Attributes** from the *Attributes* toolbox located at the top of the application window, change the *Active Level* to **SidewalkFront**.
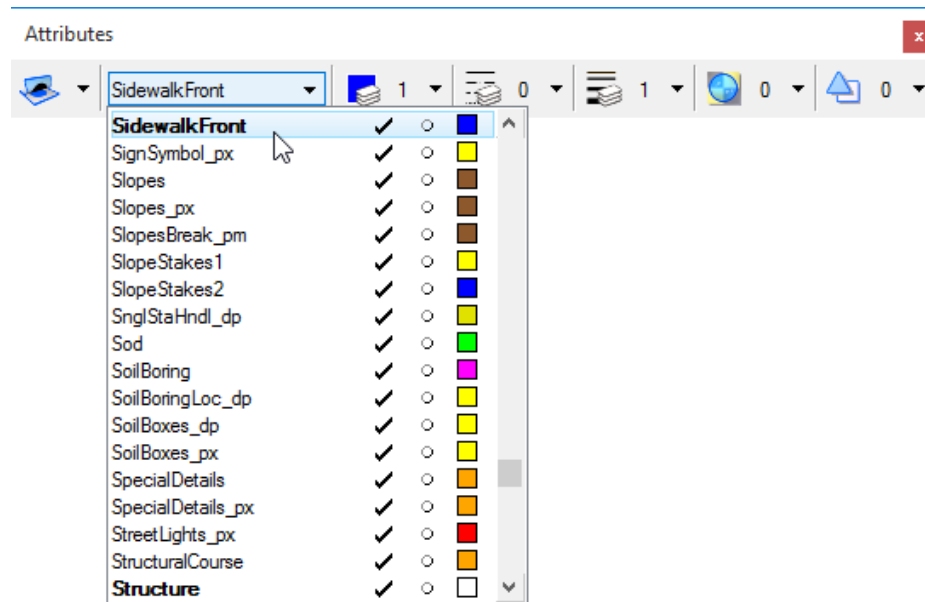
4. **Select the Tool** from the Task Dialog under the Main Classic task by finding the *Drawing* task and clicking the **Place Line** tool. The *Place Line* Tool Settings box displays.
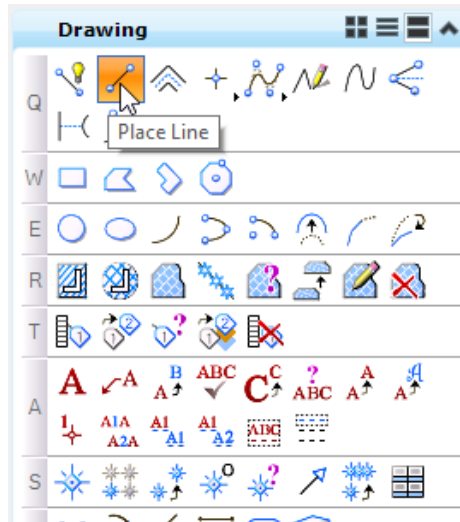
Note   This is displayed in List Layout Mode. There may be a different Layout Mode (Icon or Panel) which can easily be toggled using the icons right of the Main Classic title.

5. **Adjust the Tool Settings** from the *Place Line* Tool Settings box by selecting the *Length* checkbox and enter **200** as shown below.
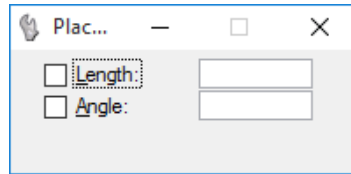
6. **Follow the MicroStation Status Bar Prompts** at the bottom left of the application window to place a line along the shoulder along the left side of the roadway between **Stations 206+00 and 208+00**.

    a. *Place Line > Identify start of line* – place a data point to start the line at **Station 206+00**.

    b. *Place Line > Identify end of line* –place another data point to end the line at **Station 208+00**.

7. Right click on the mouse to reset and end process.

8. Repeat steps 3 – 7 to set the Back of Sidewalk. From the Attributes toolbar, change the *Active Level* to **SidewalkBack** and place another line offset from the front of the side walk.
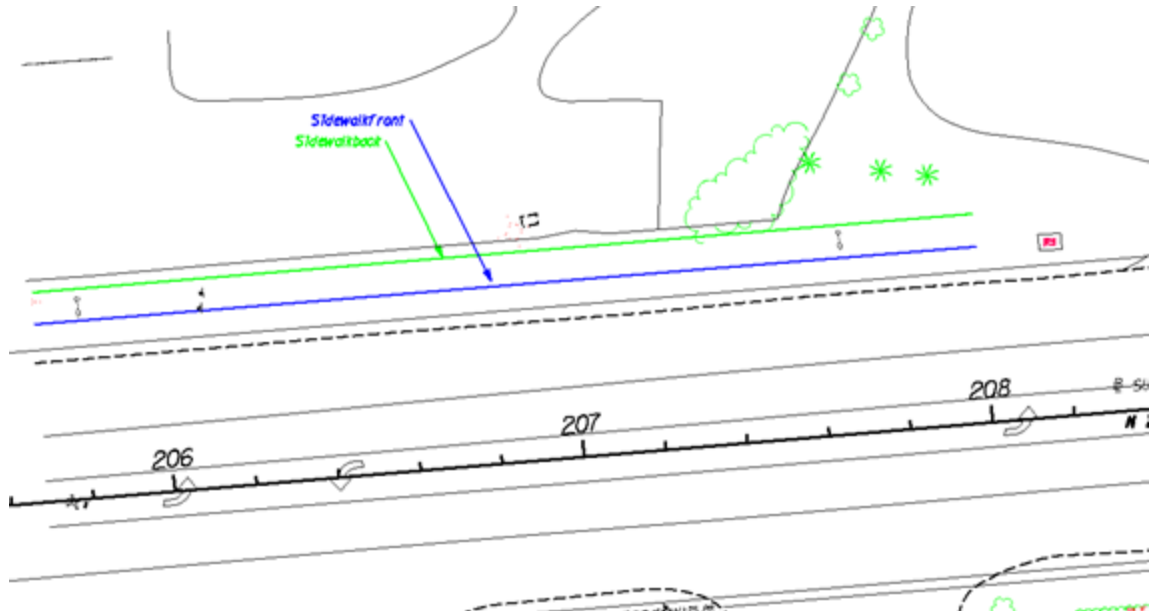
POLYGONS TASK

While it is possible to create closed polygonal shapes using the **Place Line** or **Place SmartLine** tools, it is often easier to use the tools contained in the Polygons toolbox. The Polygons toolbox contains tools that facilitate the creation of both regular and irregular polygons.
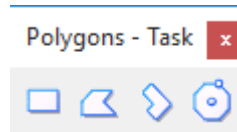
	Place Block	Used to place a block (rectangular shape).
	Place Shape	Used to place a polygonal shape by entering a series of data points to define the vertices.
	Place Orthogonal Shape	Used to place a non-rectangular shape in which each segment is either perpendicular or parallel to all other segments.
	Place Regular Polygon	Used to place a regular polygon (one with all sides and angles the same).

Note   Because polygons are closed objects, they can be created with a user-selected fill color. For the fill color to be visible, the Fill checkbox must be populated in the View Attributes dialog.

Tool settings for placing polygons vary depending on the type of shape being placed. However, each type of polygon allows setting of the *Area, Fill Type* and *Fill Color.*
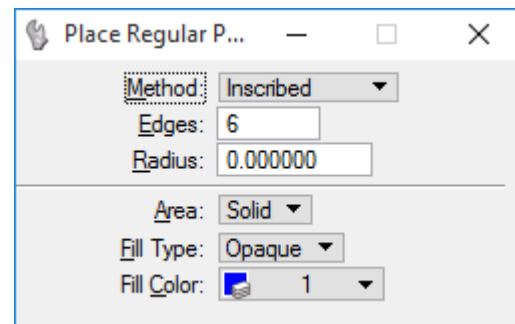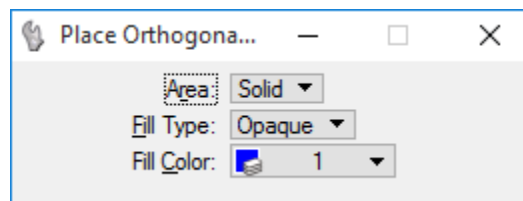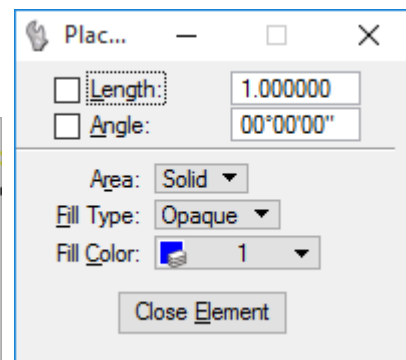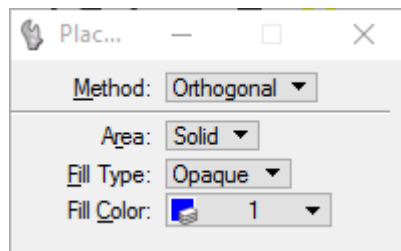
PLACING POLYGONS USING PLACE SMARTLINE

The **Place SmartLine** tool in the Linear Elements toolbox affords more flexibility to place polygonal shapes than the **Place Shape** tool, particularly when used in conjunction with AccuDraw.

The **Place SmartLine** tool is used to automatically place a complex chain or shape containing a dynamically specified collection of segments and vertices. This tool is designed to be used in conjunction with AccuDraw. The main benefits of using
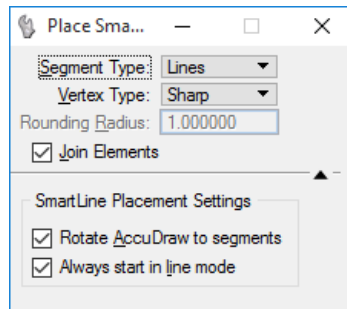
AccuDraw with **Place SmartLine** are:

- The drawing plane origin automatically moves to the location of a newly defined vertex.

- The drawing plane automatically rotates to align with a newly defined segment. This allows tangent and perpendicular segments to be easily defined.

- When defining an arc segment, the drawing plane coordinate system automatically switches to Polar.

Exercise 6.2 *Place Orthogonal Shape*

1. Continuing in the same area from the last exercise, from the *Attributes* toolbar, change the *Active Level* to **Building**.
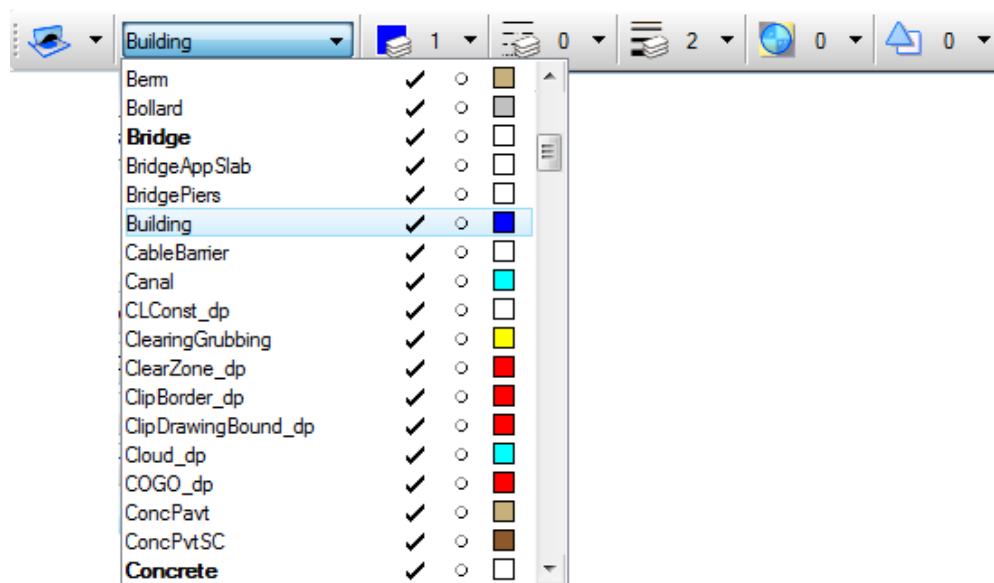
2. From Task Dialog under the Drawing task under the *Polygons* toolbar, click the **Place Orthogonal Shape** tool.
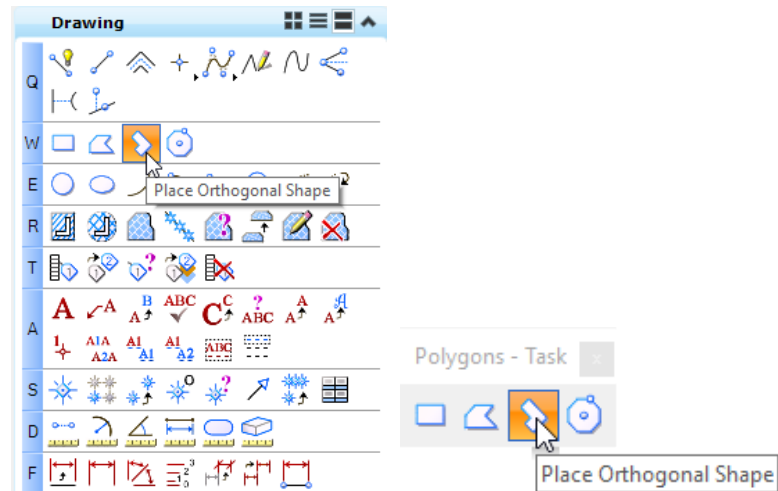
3. Set the P*lace Ortho*gonal Shape **Tool Sett***ings as show*n be**low.**
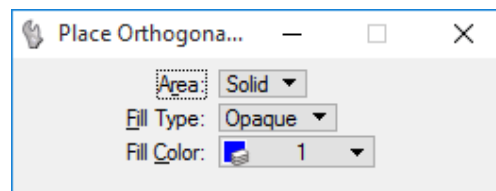
4. Following the Status Bar prompt to *Enter shape vertex*, place a **Data point** at one of the corner locations of the building as shown in the image below. Continue placing **data points** until the building is complete.
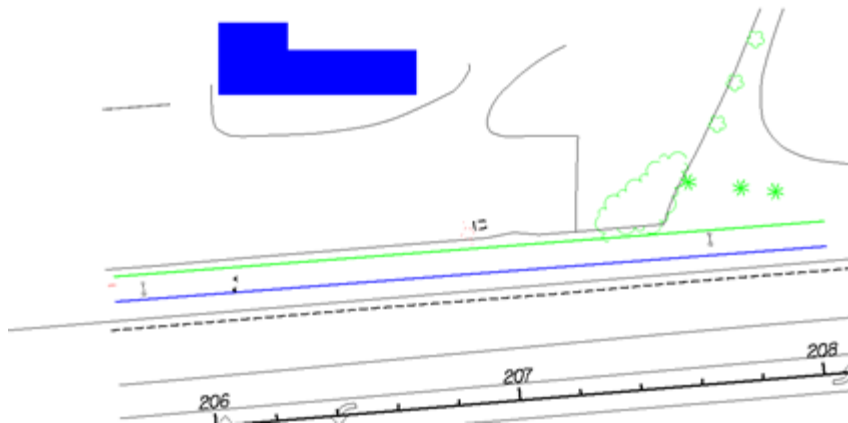
> *Note*   This tool draws all consecutive lines perpendicular and parallel beginning from the first line. When closing the shape simply snap to the beginning element and MicroStation adjusts the last line to keep the shape square.

5. When finished, **Exit** MicroStation.

CIRCLES TASK

While a straight line is one of the most common geometry types, lines alone are not enough to fully describe everyday geometric shapes. Circles and Arcs create curved segments of geometry. Out of the 8 Circles tools only five are for geometry creation. Even though there are only a few tools, many arcs can be created by manipulating the *tool settings*.
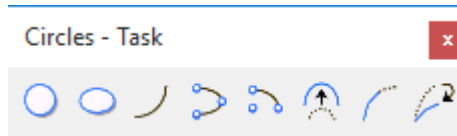
	Place Circle	Used to place a circle.
	Place Ellipse	Used to place an ellipse, precisely positioning the center and one end of the primary axis.
	Place Arc	Used to place a circular arc. Arcs can be placed clockwise or counterclockwise.
	Place Half Ellipse	Used to place an elliptical arc with a sweep angle of 180°.
	Place Quarter Ellipse	Used to place an elliptical arc with a sweep angle of 90°.
	Modify Arc Radius	Used to modify a circular arc's radius, sweep angle, and center. The endpoints do not move.
	Modify Arc Angle	Used to extend or shorten the length (sweep angle) of an arc.
	Modify Arc Axis	Used to lengthen or shorten an axis of an arc.

The **Place Circle** tool is one of the most common tools and has many important settings that can be changed in the *Tool settings* dialog. These settings control the method to place the circle.
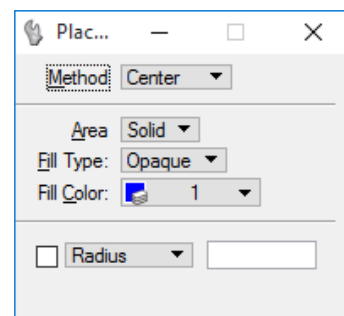
The **Place Arc** *tool settings* provide ample control for how an arc is to be placed. As with linear elements, any, all or none of the options may be used.
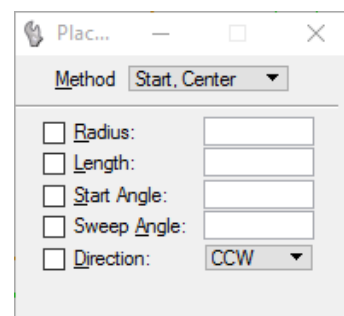
UNDO AND REDO

No matter how meticulous a user is, errors occur when creating a design. MicroStation stores commands in an **Undo** buffer that goes back a single step or multiple steps when a mistake is made. Redo is the opposite of undo.

There is no limit to the number of **Undos** that can be performed. After making the selection to **Undo** a drawing operation, the operation previous to the negated operation can be undone. A series of previous operations can be undone by repeatedly choosing Undo. Undo function cannot be used for settings changes.

More advanced undo functions can be accessed through the MicroStation menu option *Edit*:

Edit	Element	Settings
↶ Undo		Ctrl+Z
Undo Other		▶
↷ Redo		Ctrl+R
Set Mark		
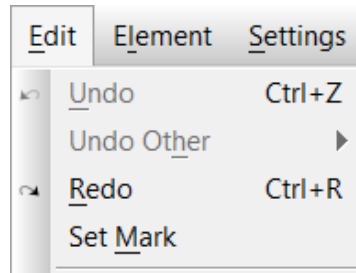
Undo	Negates the last drawing operation. Identifies the last *placed* element in the menu option.
Undo Other > All	Opens an alert box that lets the user negate all of the drawing operations recorded in the Undo buffer.
Undo Other > To Mark	Negates drawing operations performed after a mark was set by choosing Set Mark from the Edit menu.
Redo	Negates the last Undo operation. Identifies the last *undone* element in the menu option.
Set Mark	Sets a mark in the undo buffer back to which subsequent drawing operations can be undone.

> *HINT*   Each undo operation is a single re-doable operation regardless of the number of negated drawing operations.

Note   Any Compression done on the design will delete the Undo/Redo buffers.

7 DRAWING WITH PRECISION

OBJECTIVES

This chapter reviews the effects of using locks, snaps, AccuSnap and AccuDraw to control precision drafting. The topics include:

- Locks
- Snap Modes
- AccuDraw
- AccuDraw Shortcut Key-ins

INTRODUCTION

The tools discussed in this chapter are used in conjunction with drawing tools to create drawings with greater precision. These tools control accurate placement of elements by locking or constraining various settings.

LOCKS

Locks are drawing assistance tools that can be used to help produce drawings more effectively. They are settings that allow more control of the actions of various tools. These locks, to different degrees, control how the MicroStation controls the cursor, data point accuracy or groups of elements.

Note   These locks should not be confused with the element lock function. This function, accessed through Edit > Lock, allows the user to lock selected elements. The attributes and locations of locked elements cannot be changed, preventing accidental element manipulation. If an attempt is made to change a locked element, MicroStation will alert the user.

The Locks may be accessed from the MicroStation menu option **Settings > Locks**

<OR> From the MicroStation *Status Bar* at the bottom of the applications window as shown below.
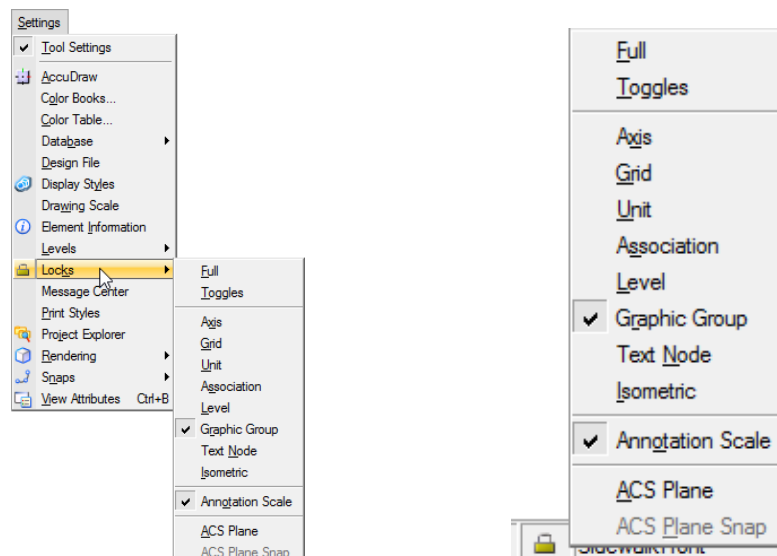
LOCKS DIALOG

The Locks dialog is used to set each of the locks and the Fence Selection Mode. It is accessed from the MicroStation menu option **Settings > Locks > Full** or Status Bar Locks popup menu option **Full.** Each Lock is enabled by toggling ON the adjacent checkbox and settings selected to effect the use of each lock.

Each lock performs a different function by controlling or constraining some portion of MicroStation. A description of each lock is shown below.
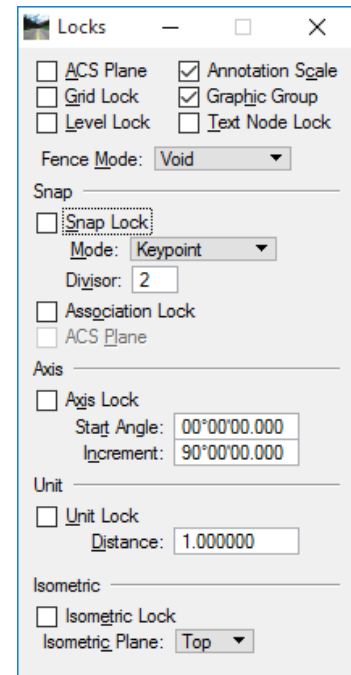
ACS Plane	(3D only) When on, each data point is forced to lie on the Active ACS's xy plane (z=0). The Active ACS is set in the Auxiliary Coordinates dialog, which is opened by choosing Auxiliary Coordinates from the Utilities menu.
Grid Lock*	When on, each data point is forced to lie on the grid.
Level Lock	When on, only elements on the Active Level can be selected; elements not on the Active Level cannot be manipulated.
Annotation Scale Lock*	When on, the annotation scale is applied to any text or dimensions that are placed in the model. The annotation scale is set through the Model Properties dialog.
Graphic Group Lock	When on, any manipulation performed on one member of a graphic group is automatically performed on all members. If off, the member elements of a graphic group can be manipulated individually.
Text Node Lock*	When on, newly placed text is attached to empty text nodes, and text cannot be placed if there aren't any empty text nodes.
Fence Mode	Sets how the fence contents are determined for element manipulations. Fence (Selection) Mode is also a tool setting for the fence placement tools in the Fence toolbox.
Snap Section	The Snap section contains controls that are used to determine how tentative points snap to elements.
Axis Section	The Axis section contains controls that are used to adjust Axis Lock and related settings.
Unit Section	The Unit section contains controls that are used to adjust Unit Lock and a related setting.
Isometric Section	The Isometric section contains controls that determine how data points relate to the Isometric Plane.

* These locks are not used by FDOT.

LOCKS TOOLBOX

The Locks toolbox is available to allow the user to turn locks on and off. Clicking the tool has the same effect as selecting the check boxes in the Locks dialog. It can be accessed from the MicroStation menu option **Settings > Locks > Toggles** or Status Bar Locks popup menu option **Toggles.** This may be beneficial when constantly switching back and forth between multiple lock settings.
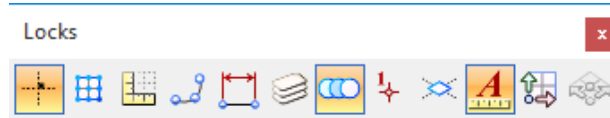
SNAP MODES

When Snap Lock is on, how a tentative point snaps to an element is set by the active *Snap Mode* (or the override setting if one is active). The default Snap Mode can be set via the Locks dialog, Snap Mode button bar, pointer's pop-up menu or Status Bar. The active Snap Mode (or Snap Mode override) is indicated by the icon in the Status Bar. The Snap Mode can be access from the MicroStation menu option **Settings > Snaps** or Status Bar Snaps popup menu**.**
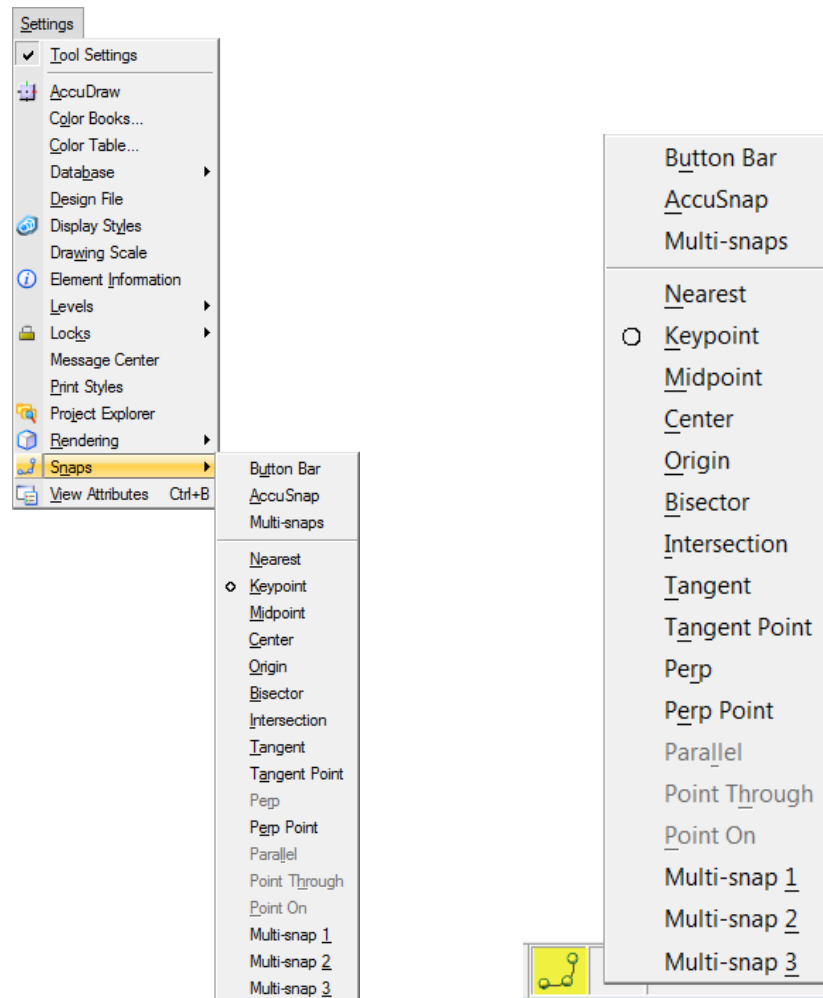
To override the snap mode, select the **Active Snap Mode** icon on the Status Bar and then click a snap mode in the pop-up menu. This temporarily overrides the default snap mode for one operation, after which the override is canceled and the original snap mode becomes active again. A solid dot indicates an override.

SNAP MODE BUTTON BAR

With Snap Lock on, each snap mode setting has an effect on tentative snap points. Using the Snap Mode button bar, a single click on a button overrides the current snap mode for one operation. There is one button for each type of snap and a button [icon] to turn AccuSnap on and off. Only the buttons that are appropriate for the Active Tool will be enabled for use at any given time. All others that are contextually inappropriate are disabled (dimmed)

The Snap Mode button bar is accessed by selecting **Button Bar:**

from the **Active Snap Mode** icon on the Status Bar

<OR> from the MicroStation menu option **Settings > Snaps > Button Bar**

<OR> from the FDOT Menu option **Actions > Snap Mode Button Bar**.

Snap Modes

[icon]	**Near Point**	Snaps to the closest points on elements.
[icon]	**Keypoint**	Snaps to the pre-defined keypoints on elements (endpoint, midpoint and radius).
[icon]	**Midpoint**	Snaps to the midpoints of elements and segments of elements.
[icon]	**Center**	Snaps to the center of elements (such as circles, arcs, text, etc.) with centers and the centroid of other elements, including shapes, line strings, and B-splines.
[icon]	**Origin**	Snaps to the origin of a cell or text, centroid of a B-spline, the first data point in a dimension element, or the first vertex of a line, multi-line, line string, or shape.
[icon]	**Bisector**	Snaps to the midpoints of entire element.
[icon]	**Intersect**	Snaps to the intersection of two elements. Two tentative points are required, although more can be used. The first tentative point snaps to one element, and that element is highlighted. The second tentative point snaps to another element, and the two segments used to find the intersection of the two elements are displayed in dashed lines. If the two elements do not actually intersect, but projections of the elements would intersect, the segments include projections of the elements to the intersection. Continue snapping until the desired intersection is found; the last two tentative points define where the intersection snap lies.
[icon]	**Tangent**	The edge of the element being placed is constrained to be tangent to an existing element. The tentative point dynamically slides along the element to maintain the tangency as the pointer moves to finish placing the element.
[icon]	**Tangent Point**	The edge of the element being placing is constrained to be tangent to the existing element at the tentative point. The tentative point does not move dynamically as the pointer moves, but is locked in place.

	Perpendicular	The line being placed is constrained to be perpendicular to the element. The tentative point slides dynamically along the element in order to maintain the perpendicularity as the pointer moves to finish placing the element.
	Perpendicular Point	The line being placed is constrained to be perpendicular to the element at the tentative point. The tentative point does not move dynamically as the pointer moves, but is locked in place.
	Parallel	Does not define a point through which the line being placed passes. Instead, when the tentative point is accepted, the line placed is parallel to the line to which the tentative point was snapped.
	Through Point	Defines a point through which the element being placed must pass.
	Point On	Constrains an element to begin or end on an element in the design file.
	Multi-snap1	By default is Intersection, Keypoint, Nearest.
	Multi-snap2	By default is Intersection, Keypoint, and Center.
	Multi-snap3	By default is Midpoint, Intersection, and Center.

Tip *If elements are overlapped, select the desired element with Element Selection tool and then select Edit > Bring to Front from the Menu Bar. This brings the element to the top and makes it easier to snap to.*

MULTI-SNAPS DIALOG

The Multi-Snaps dialog is used to set up groups of snaps known as multi-snaps. When multi-snaps is active, AccuSnap and Tentative Point snap processes the list of snaps that it represents. The order in which the snaps are processed can be changed by dragging and dropping the list entries.

The Multi-Snap dialog is accessed by:

selecting **Multi-Snaps** from the **Active Snap Mode** icon on the Status Bar

<OR> from the MicroStation menu option **Settings > Snaps > Multi-Snaps**

<OR> from FDOT Menu option **Actions > Multi-Snap Dialog**.

The icon bar is used to set the multi-snap mode to define.

- Multi-snap 1 — By default: Intersection, Keypoint, Nearest.
- Multi-snap 2 — By default: Intersection, Keypoint, Center.
- Multi-snap 3 — By default: Midpoint, Intersection, Center.
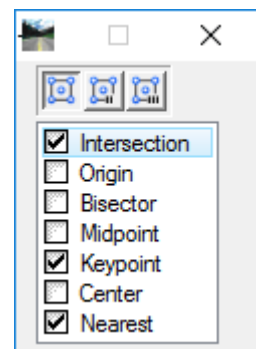
ACCUSNAP

AccuSnap enhances many of the standard snap mode settings by displaying and automatically snapping to the next tentative snap point as the pointer is moved over an element. With AccuSnap enabled, a tentative snap point is very rarely needed to be placed manually.

Note   Tentative snap points can still be entered manually with AccuSnap enabled.

AccuSnap can be toggled on or off by clicking the toggle **AccuSnap** button on the Snap Mode button bar located at the far left.

AccuSnap works in conjunction with the current Snap Mode setting and, for the most part, is similar in operation to the manual method of tentative snap points — minus the button presses. How close to an element, or a keypoint, the pointer must be before AccuSnap finds it is governed by the Locate Tolerance setting in the Input category of the Preferences dialog.

The AccuSnap Settings dialog contains a series of controls that affect the behavior of AccuSnap. Controls in this dialog are grouped into three tabbed sections; *General*, *Elements*, and *Feel*. To access the AccuSnap Settings dialog, select the MicroStation menu option **Settings > Snaps > AccuSnap** or click on the **AccuSnap** option from the Active Snap Mode pop-up menu on the Status Bar.
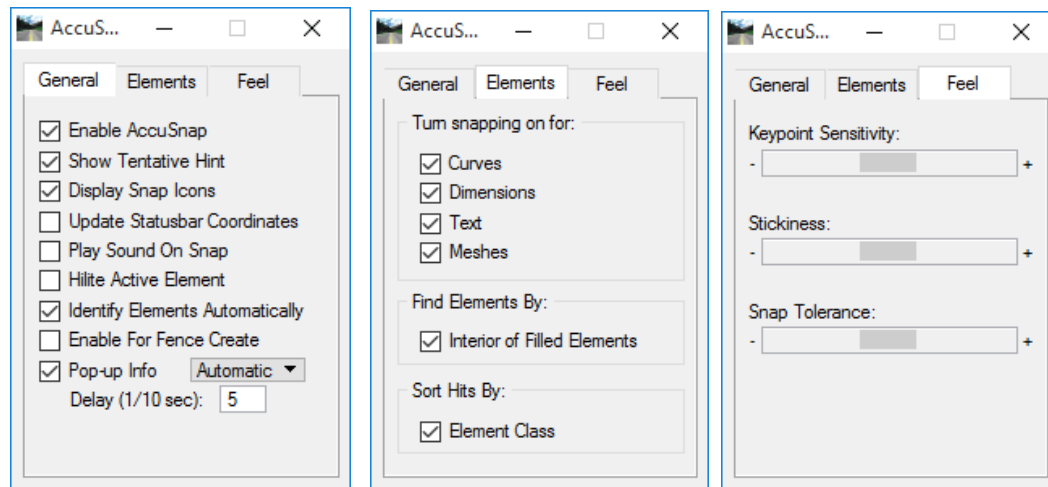
- **AccuSnap Settings General Tab**

The General tab contains controls to enable/disable AccuSnap, and to define the way it operates.

- **AccuSnap Settings Elements Tab**

This group controls whether or not AccuSnap snaps to Curves, Dimensions, Text, and/or Meshes. When snapping is turned off for any of these, AccuSnap will not snap to the particular element type, but it will display an icon to show that the element is being ignored.

- **AccuSnap Settings Feel Tab**

Using the controls in the Feel group of the AccuSnap settings allows for setting AccuSnap sensitivity when snapping to elements.

ACCUDRAW

AccuDraw is a drafting aid that evaluates such parameters as the current pointer location, the previously entered data point, the last coordinate directive, the current tool's needs, and any directive entered via shortcut key-ins or AccuDraw options. AccuDraw then generates the appropriate precision coordinates and applies them to the active tool.

Because of its versatility, the AccuDraw function reduces the need to rely on other drawing assistance tools such as grids or locks. Nearly all drawing and editing techniques can be performed faster using AccuDraw than any other drawing aid.

AccuDraw has two main areas; the AccuDraw Window and the AccuDraw Compass. The AccuDraw Compass gives visual feedback as elements are added or modified. The AccuDraw Window is used to enter precision key-ins.

By default, AccuDraw is activated automatically when MicroStation is started. With AccuDraw activated, all drawing tools utilize it for dynamic data input.

ACCUDRAW WINDOW

The AccuDraw window opens or closes when clicking the AccuDraw icon on the Primary Tools toolbox. If using the default function key menu, pressing <F11> opens the AccuDraw window. If it is already docked or open, pressing <F11> sends focus to the AccuDraw window.

Once opened, the AccuDraw window automatically takes the focus whenever dynamic update occurs with a drawing tool selected. The AccuDraw window is dockable.
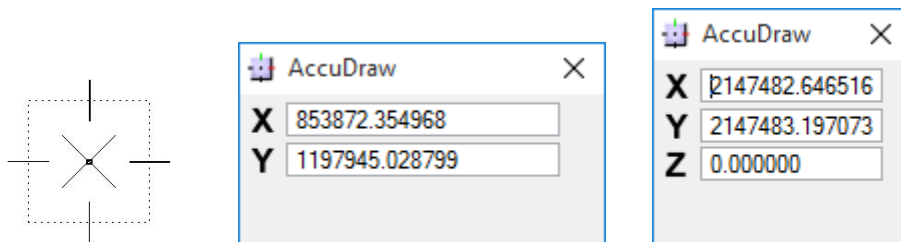
Rectangular coordinate system
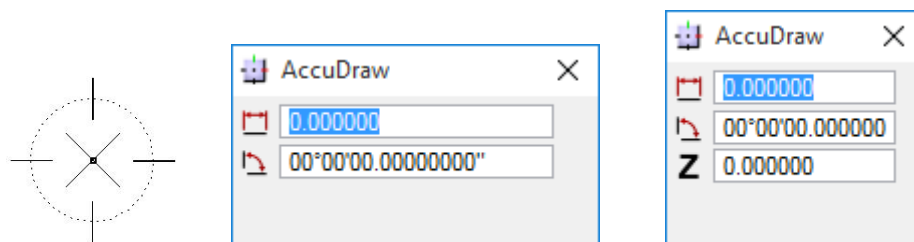
Polar coordinate system

The fields in the AccuDraw window depend on the type of drawing plane coordinate system is in effect. Rectangulate coordinates specify distances along the drawing plane axes X, Y, and (Z in 3D). Polar coordinates specify distances and angles relative to the drawing plane origin and Z value in 3D.

ACCUDRAW COMPASS

The most recognizable AccuDraw feature is its compass. Visible only when AccuDraw is active and in control of MicroStation coordinate input, the compass acts as both a status indicator and a focus for input. Normally, the AccuDraw compass does not appear until the first data point is placed after selecting a tool. AccuDraw behavior changes slightly with tools that utilize dynamics before the first data point. The *Place Text* and *Place Active Cell* tools are examples of such tools. Instead of "waiting" for the first data point to display the compass, AccuDraw activates the compass at the last data point location. In this way, the element can be placed with respect to this location by using any of AccuDraw relative directives.

The shape of the compass indicates the type of drawing plane coordinate system is in effect. Rectangular coordinate compass displays with a square shape. Polar coordinate compass displays with a circle shape. When AccuDraw has focus, its compass has a transparent blue background. When it does not have focus, the compass displays grey.

To shift from Rectangular to Polar coordinates, first check that the focus is in the AccuDraw window, then press the <spacebar>, which has a toggle effect between the coordinates. Rectangular or Polar coordinates also can be set from the AccuDraw settings dialog.
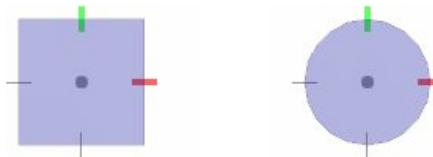
Rectangular coordinate system.   *Polar coordinate system*

THE ORIGIN POINT

At the center of the compass is the AccuDraw origin point. This is the focal point of all AccuDraw operations. As various AccuDraw options are selected, their functions operate from this origin point.

This last point is important. All AccuDraw commands (distance key-ins, shortcut key-ins, and so on) operate in conjunction with the compass. In most cases, the compass is located at the last entered data point. In addition, there are AccuDraw directives that move the compass to entirely different locations without generating a new data point.

THE FRAME (DRAWING PLANE INDICATOR)

Surrounding the origin point is the frame, or drawing plane indicator. Its primary purpose is to show the current orientation of AccuDraw' drawing plane and the current coordinate system in effect. When the frame displays as a rectangle, the rectangular coordinate system (X, Y) is in effect. When the frame displays as a circle then the polar coordinate system (Distance, Angle) is active. By default, when AccuDraw has focus, the frame's color is grey, and its fill color is blue, but this can be changed in the Display tab of the AccuDraw Settings dialog.

THE X/Y AXES

The final parts of the AccuDraw compass are the two axes. Oriented at right angles to one another, these axes represent the drawing plane X and Y axes.

To differentiate between the two axes, each is color coded. By default, the positive or +X axis is displayed in red while the positive or +Y axis is green in color. If these colors not acceptable or are hard to discern, they can be changed via the Display tab section of the AccuDraw Settings dialog. As well, the highlight color can be changed for when the pointer is indexed to the X or Y axes.

To Index the Compass Axes

- Move the pointer near one of the four axis indicators.

- Once the pointer is close enough, it snaps to the axis and highlights.

Notes:

o The AccuDraw compass rotates itself automatically when the pointer is moved to an angle. To return the compass to base rotation, type [B] on the keyboard.

o AccuDraw remembers the last distance entered and accepted. When the pointer is near the previous distance, a small perpendicular segment displays at the end of the line. Data point to accept the previous distance.

o AccuDraw works with all drawing and modifying tools.

o The easiest way to use AccuDraw is to move the pointer in the desired direction, type the desired distance, and then data point.

o Pressing the [Spacebar] changes the AccuDraw compass from rectangular coordinates to polar coordinates.

o Coordinates entered into the AccuDraw window are relative to the current origin point.

USING ACCUDRAW

Most of the time, a combination of mouse movements and keyboard entry is required to place elements with AccuDraw. It is not necessary to delete existing information in the AccuDraw window to input new information. To move from one input field to another, press the **[Tab]** key <OR> the up/down arrow key.

ACCUDRAW AND THE TENTATIVE POINT

Creating precision offsets is a fundamental function of the design process. AccuDraw performs this important function by utilizing tentative points. These can be input manually, or the process can be further streamlined using AccuSnap.

Whether using AccuSnap or manually entered tentative points, the key to using AccuDraw and the tentative point is the Set Origin function (the <O> shortcut key-in). When there is a tentative point present, pressing the <O> key results in the relocation of the compass to the tentative point location.

AccuDraw enters a dynamic tentative point mode where the final data point location can be manipulated as follows:

• Enter coordinate data using the AccuDraw window.

• Use a second tentative point and various locks (X/Y or D/A) to interactively set the offset.

To Create an offset from an element, along the x or y axis:

1. With a tool active, use AccuSnap to display a tentative point at the required point on the element (or place it manually).

2. Press the <O> shortcut key. The compass moves to the tentative point location.

3. Move the pointer so that it is indexed on the required X or Y axis. This sets the focus automatically to the indexed axis.

4. Key in the required distance.

5. With the pointer still indexed to the axis, accept with a data point.

Locking the X or Y values

Another method of aligning with existing elements is to snap to the desired location and lock the axis. The **[X]** and **[Y]** shortcuts lock the X and Y axes respectively, and **[D]** and **[A]** lock the distance and angle.

1. With a tool active, use AccuSnap to display a tentative point at the keypoint of the first element (or place it manually).

2. Press the <O> shortcut key. The compass moves to the tentative point location.

3. On the keypoint of the second element, use AccuSnap to display a tentative point (or place it manually).

4. Press the <X> or <Y> key. AccuDraw responds by locking the dynamic line to the appropriate X or Y axis, but at the offset from the first tentative point to the second one.

5. Use another tentative point to "lock" the remaining axis of the tentative compass location. <or> Enter a data point to set the second axis value. AccuDraw relocates the initial compass to the new indirectly derived location. From here the use of the current drawing tool can continue.

Using the Smart Lock

Another method is to toggle on the Smart Lock tool. It is activated by pressing the **[Enter]** key on the keyboard. Smart Lock locks the axis that currently has AccuDraw focus.

ACCUDRAW POPUP CALCULATOR

It is often useful to enter not only a distance, but the total length of many distances, or perhaps a distance minus a certain amount. AccuDraw has a built-in calculator to handle many of these circumstances.

The AccuDraw Pop-up Calculator performs standard mathematical operations, such as addition, subtraction, multiplication and division, on the values displayed in some MicroStation text fields and the values adjust accordingly. Its use is extendable to support other controls, such as, Active Scale and Active Angle fields.

To use the Pop-up calculator

1. With the input focus in a field that supports pop-up calculation, type any of the following:

+	Addition Operator
-	Subtraction Operator
∗	Multiplication Operator
/	Division Operator
[Enter]	Accepts Calculation

2. Type a value or expression to complete the calculation.

3. Press <Enter>, enter a data point, or click outside the pop-up calculator to accept the calculated value.

4. Press <Esc> if rejecting the value.
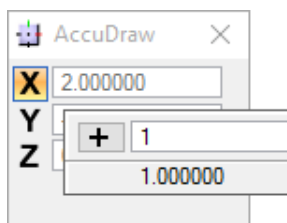
ACCUDRAW SETTINGS

AccuDraw is a powerful drawing construction tool, and it has many associated control settings that are used to select preferences for operation and alter aspects of the way AccuDraw performs. Most of these settings are contained in the AccuDraw Settings dialog under three tabs to group the settings in an intelligent way. The AccuDraw Settings dialog can be accessed through the MicroStation menu options **Settings > AccuDraw**.

OPERATION TAB
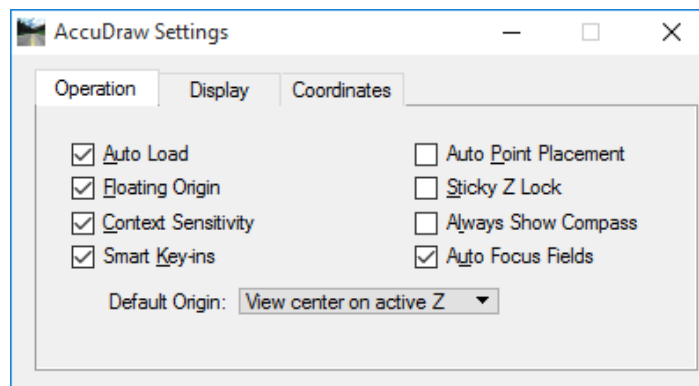
|	
Auto Load	If on (the default), AccuDraw starts automatically when MicroStation is launched.
Floating Origin	If on (the default), the origin moves to the last point placed.
Context Sensitivity	If on (the default), enables tools to provide "hints" to AccuDraw to override its default behavior for smoother operation. This may include setting custom "Context" rotations, the origin, the coordinate system type and locking values.
Smart Key-ins	If on (the default), AccuDraw interprets a number as positive or negative, depending on the direction of the pointer from the compass. In rectangular mode only, the Smart Key-ins will cause AccuDraw to move the focus to either the x or the y field depending on pointer position.
Auto Point Placement	If on, places data points automatically when they are fully constrained (if both the X and Y values are locked, or if one or the other is locked while the pointer is indexed to zero). By default, this option is off.
Sticky Z Lock	If on, when the Z axis is locked, it remains locked through consecutive operations. Normally, locks are cleared when the user enters a data point.
Always Show Compass	If on, when AccuDraw is active, the compass displays prior to placing a data point for the current operation.
Auto Focus Fields	(Polar coordinates only) If on, the <A> and <D> AccuDraw shortcuts set focus to the selected field and lock the current value. If off, the <A> and <D> AccuDraw shortcuts lock the current value without affecting the focus.
Default Origin	Lets the user choose the default origin. When a tool starts AccuDraw and there is no origin currently defined, then this setting specifies the default location of the AccuDraw drawing plane origin.

DISPLAY TAB
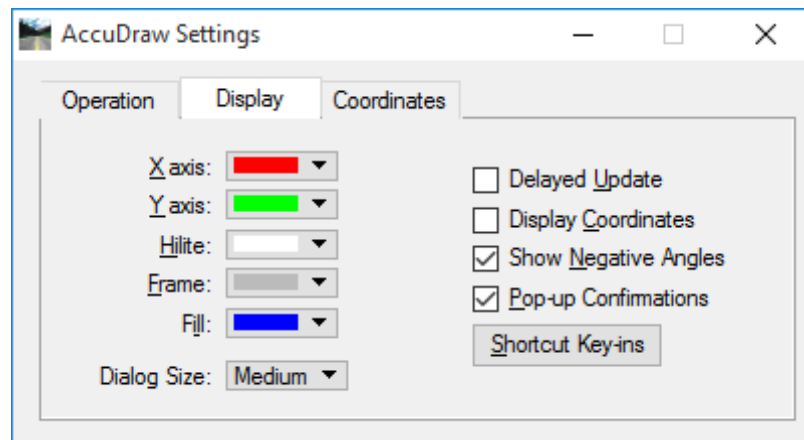
X Axis	Changes, via an option menu, the color for the positive X axis indicator on the AccuDraw compass.
Y Axis	Changes, via an option menu, the color for the positive Y axis indicator on the AccuDraw compass.
Hilite	Changes, via an option menu, the color for the negative X and Y axis indicators on the AccuDraw compass.
Frame	Changes, via an option menu, the color for the AccuDraw compass frame.
Fill	Sets the fill color of the inside of the AccuDraw compass.
Dialog Size	Sets the width of the AccuDraw window.
	Small — 100 pixels
	Medium — 126 pixels
	Large — 152 pixels
Delayed Update	If on, X/Y coordinates are updated in the AccuDraw window when the cursor comes to a rest. If off (the default), coordinates are updated in the AccuDraw window continuously, as the cursor is moved.
Display Coordinates	If on, a coordinate display at the pointer shows the increments in the x, y, and z (for 3D) directions from the previous data point.
Show Negative Angles	If on (the default), AccuDraw displays negative angles (that is, +/-180°). If off, AccuDraw displays angles as 0° to 360°.
Pop-up Confirmations	If on (the default), AccuDraw displays shortcut pop-ups.
Shortcut Key-ins	Opens the AccuDraw Shortcuts menu that lists the AccuDraw shortcut keys.

COORDINATES TAB
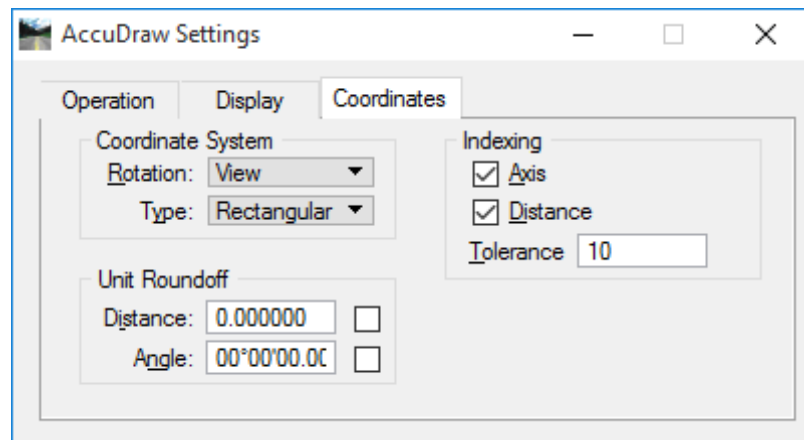
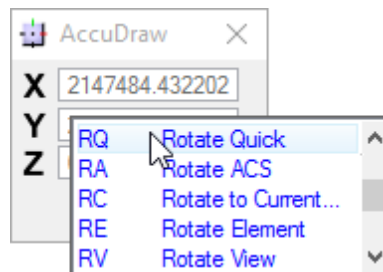|	
Coordinate System	Contains controls used to rotate an AccuDraw drawing plane and set an AccuDraw Coordinate System.
Rotation	Sets the rotation of the drawing plane axes, via an option menu. While the AccuDraw window has the focus, pressing the key combination indicated by the underlined letters has the same effect as choosing the specified Rotation, i.e. Entering RV = Rotation: View.
Type	Sets the method used in the AccuDraw window to specify distances and angles relative to the drawing plane origin.
Unit Roundoff	Contains controls used to set the round-off value for distances and angles relative to the drawing plane origin.
Distance	Sets the round-off value for distances relative to the drawing plane origin. While the accompanying check box is on, the round-off value is effective, except if overridden by keyed in values or snapped tentative points. The check box turns on automatically when the Distance setting is entered.
Angle	Sets the round-off value, in degrees, for angles relative to the drawing plane origin when in Polar mode. While the accompanying check box is on, the round-off value is effective except if overridden by keyed in values or snapped tentative points. The check box turns on automatically when the Angle setting is entered.
Indexing	Contains controls that control AccuDraw indexing functionality.
Axis	If on, indexing is activated for AccuDraw drawing plane X and Y axes. The distance required to move the pointer from the indexed axis is controlled by the *Tolerance* setting.
Distance	If on, sets the minimum distance that the pointer must move from the AccuDraw origin in order to place a new data point. The minimum distance is controlled by the *Tolerance* setting.
Tolerance	Sets the minimum distance, in screen pixels, used by the Axis and Distance settings. Allowable values are 1–99.

ACCUDRAW SHORTCUT KEY-INS

AccuDraw actually affects the operation of most MicroStation tools, and even has the potential to work with third-party application software. Although AccuDraw tries to anticipate the next move it cannot always predict intentions. AccuDraw therefore includes a wide variety of single and double character command directives known as the shortcut key-ins.

By pressing the appropriate key, AccuDraw can be directed to perform a specific task. In operation, AccuDraw is the default input focus in most cases, thus allowing only the shortcut to be typed without having to consider where the input focus is.

Pop-ups confirm single letter shortcuts below the focused input field. This function is not only useful to confirm the key-in, but also serves to communicate that the shortcut is received via AccuDraw input fields.
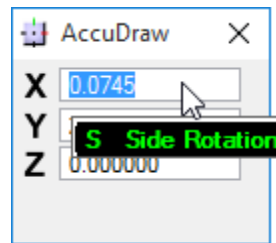
Two letter shortcuts appear attached to the focused input field, and confirm the action through the temporary appearance of the AccuDraw Shortcuts window. The pop-down list adjacent to the input field illustrates where the shortcut comes from, thereby making it easy to "follow the action."

The following table lists each keyboard shortcut and its effect.

Key	Effect
\<Enter\>	Smart Lock

- In Rectangular coordinates, locks the X field to 0 if the pointer is on the drawing plane y-axis or the Y field to 0 if the pointer is on the x-axis.

- In Polar coordinates, locks Angle to 0°, 90°, -90°, or 180° if the pointer is on a drawing plane axis; otherwise, locks Distance to its last entered value.

\<Space Bar\>	Toggles between Rectangular and Polar coordinates.
\<?\>	Opens the AccuDraw Shortcuts window.
\<~\>	Bumps an item in the *Tool Settings* dialog (there is no need to press the **[Shift]** key). It finds the first enabled toggle button or an option button in the *Tool settings* dialog and either toggles it or bumps it to the next valid value.
\<O\>	Moves the drawing plane origin to the current pointer position. This can also be used to explicitly activate AccuDraw before entering a data point.
\<V\>	Rotates the drawing plane to align with the view axes. Pressing this key a second time restores context-sensitive rotation.

\<T\>	Rotates the drawing plane to align with the axes in a standard Top view (3D only). Pressing this key a second time restores context-sensitive rotation.
\<F\>	Rotates the drawing plane to align with the axes in a standard Front view (3D only). Pressing this key a second time restores context-sensitive rotation.
\<S\>	Rotates the drawing plane to align with the axes in a standard Side view (3D only). Pressing this key a second time restores context-sensitive rotation.
\<B\>	Rotates the drawing plane to align with the active ACS, or if a rotation is set up in the dialog, it returns to that rotation. In a new file (where ACS hasn't been used yet) it is the rotation of the view.
\<E\>	Rotates between three main planes: top, front, and side (3D only). This also works when the original plane is an ACS or context rotation, so the user does not have to use **[R]** + **[X]**, **[R]** + **[Y]** to rotate to a 90° plane.
\<X\>	Toggles the lock status for the *X* value.
\<Y\>	Toggles the lock status for the *Y* value.
\<Z\>	Toggles the lock status for the *Z* value (3D only).
\<D\>	Toggles the lock status for the *Distance* value.
\<A\>	Toggles the lock status for the *Angle* value.
\<L\> + \<I\>	Locks the current index state. If an axis or distance is not indexed, indexing is disabled. On the other hand, if an axis or distance is indexed, it is locked. The effect is temporary, lasting until a data point is entered or the shortcut is run again. This is useful if the user needs to index to one axis but not the other, or to enter a data point very close to an axis, but not on the axis.
\<L\> + \<P\>	Toggles ACS Grid Plane lock, which toggles the ACS Plane and ACS Plane Snap locks, and the Grid view attribute for all views.
\<L\> + \<A\>	Toggles ACS Plane lock.
\<L\> + \<S\>	Toggles ACS Plane snap lock.
\<L\> + \<Z\>	Toggles Sticky Z Lock, which is used in conjunction with ACS Plane Snap Lock to force a series of snap points to lie on the active ACS' XY plane (Z=0).
\<R\> + \<Q\>	Used to quickly and temporarily rotate the drawing plane.
\<R\> + \<A\>	Permanently rotate the drawing plane. Because it rotates the current ACS, this rotation is still active after the tool in use is exited. When active, the tool setting *Use Current Origin* causes the drawing plane origin to be used as the x-axis origin, thereby eliminating the need to enter an extra data point.
\<R\> + \<C\>	Rotates the drawing plane to the current ACS.
\<R\> + \<E\>	Rotates the drawing plane to match the orientation of a selected element.
\<R\> + \<V\>	Rotates the active view to match the current drawing plane.
\<R\> + \<X\>	Rotates the drawing plane 90° about its x-axis.
\<R\> + \<Y\>	Rotates the drawing plane 90° about its y-axis.
\<R\> + \<Z\>	Rotates the drawing plane 90° about its z-axis.
\<G\> + \<T\>	Moves focus to the *Tool settings* dialog.

\<G\> + \<K\>   Opens (or moves focus to) the Key-in window (equivalent to selecting **Utilities > Key-in** from the Menu Bar).

\<G\> + \<S\>   Opens (or moves focus to) the AccuDraw Settings dialog (equivalent to selecting **Settings > AccuDraw** from the Menu Bar).

\<G\> + \<A\>   Opens the Get ACS dialog, which saves an Auxiliary Coordinate System.

\<W\> + \<A\>   Opens the Write to ACS dialog, which saves the drawing plane alignment as an ACS.

\<P\>   Opens the Data Point Key-in dialog for entering a single data point.

\<M\>   Opens the Data Point Key-in dialog for entering multiple data points.

\<I\>   Activates Intersect snap mode.

\<N\>   Activates Nearest snap mode.

\<C\>   Activates Center snap mode.

\<K\>   Opens the Keypoint Snap Divisor dialog, which is used to set the Snap Divisor for keypoint snapping.

\<H\> + \<A\>   Suspends AccuDraw for the current tool operation. Selecting a new tool, or entering a Reset re-enables AccuDraw.

\<H\> + \<S\>   Toggles AccuSnap on/off.

\<H\> + \<U\>   Suspends AccuSnap for the current tool operation. Selecting a new tool or entering a Reset re-enables AccuSnap.

\<Q\>   Deactivates AccuDraw.

Exercise 7.1 *Placing Additional Topo Elements*

➢ *Place A Missing Fire Hydrant*

According to the survey field notes, the fire hydrant should have been in the drawing at **Station 206+68**, **60 feet** left of centerline. Instead it is located approximately at **Station 207+00**, **70 feet** left of centerline. This exercise will relocate the fire hydrant along the left side of the roadway.

1. Open the MicroStation File c:\e\projects\12345678901\survey*topord01.dgn* and zoom into the location of **Stations 206+00 to 208+00** at the *beginning of* the project.

2. From the *Primary* to**olbar, ensure that the Ac**cuDraw is toggle **ON**.

3. Rotate the image as follows:

   a. Using the *View Controls*, select the *Rotate View* icon and set the *Method* to **2 Points**.
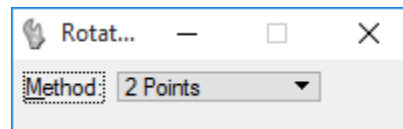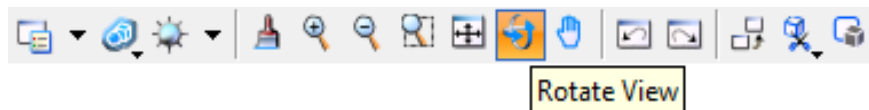
   b. As per the Status Bar prompt, *define the first point* by placing a data point on **Station 206** and then *define X axis of view* by placing a second data point on **Station 207**. The image automatically rotates along that line.
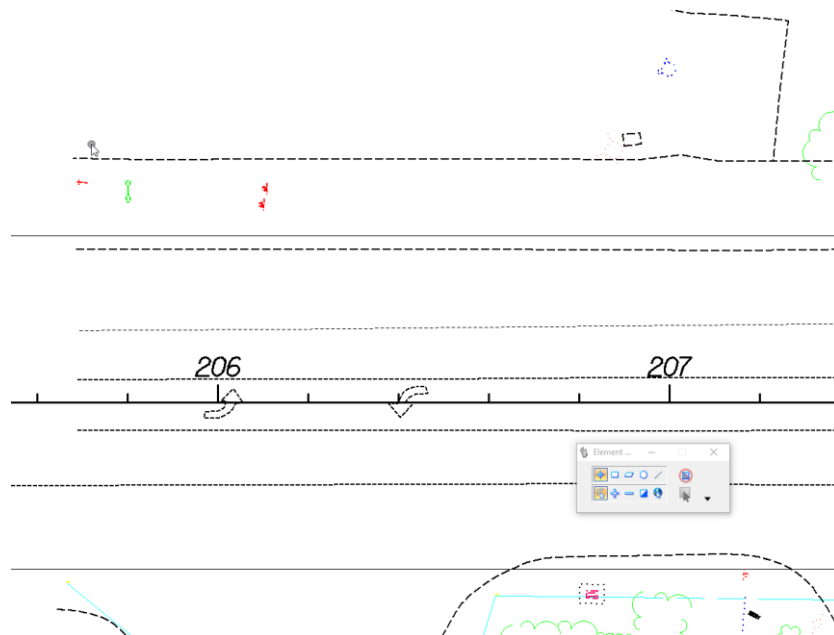
4. Do not worry about the element symbology. AccuDraw and the Place SmartLine tool are only being used to help locate the fire hydrant station and offset. The line will be deleted at the end of this exercise.

5. From the Drawing task, click the **Place SmartLine** icon.
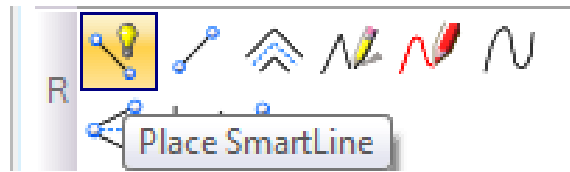

6. Place a Tentative Point at the intersection of the centerline and **Station 206+00** tic mark and then place a Data point to accept the tentative location to start the Place Sm**artLine tool.**
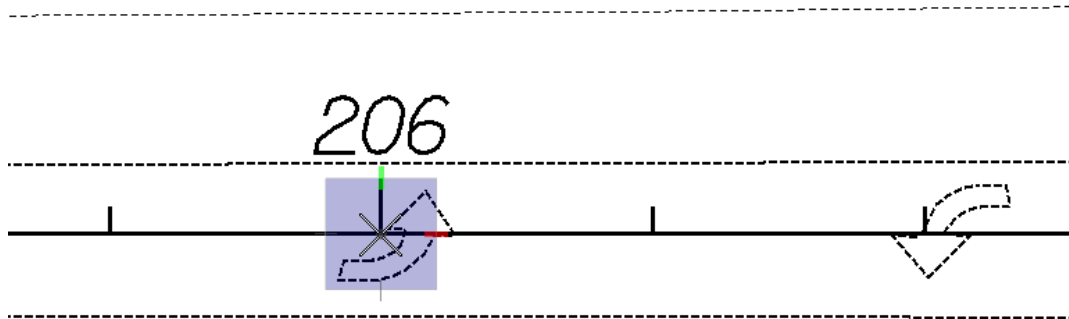

7. Press the AccuDraw shortcut keys **[R] + [Q]** (Rotate Quick).

8. Move the cursor to the right. A preview of the line that will be drawn is shown and the X Value in AccuDraw will highlight. Using the AccuDraw *SmartLock* feature as shown below, key-in a *distance* of **68.00.** The length is locked in.
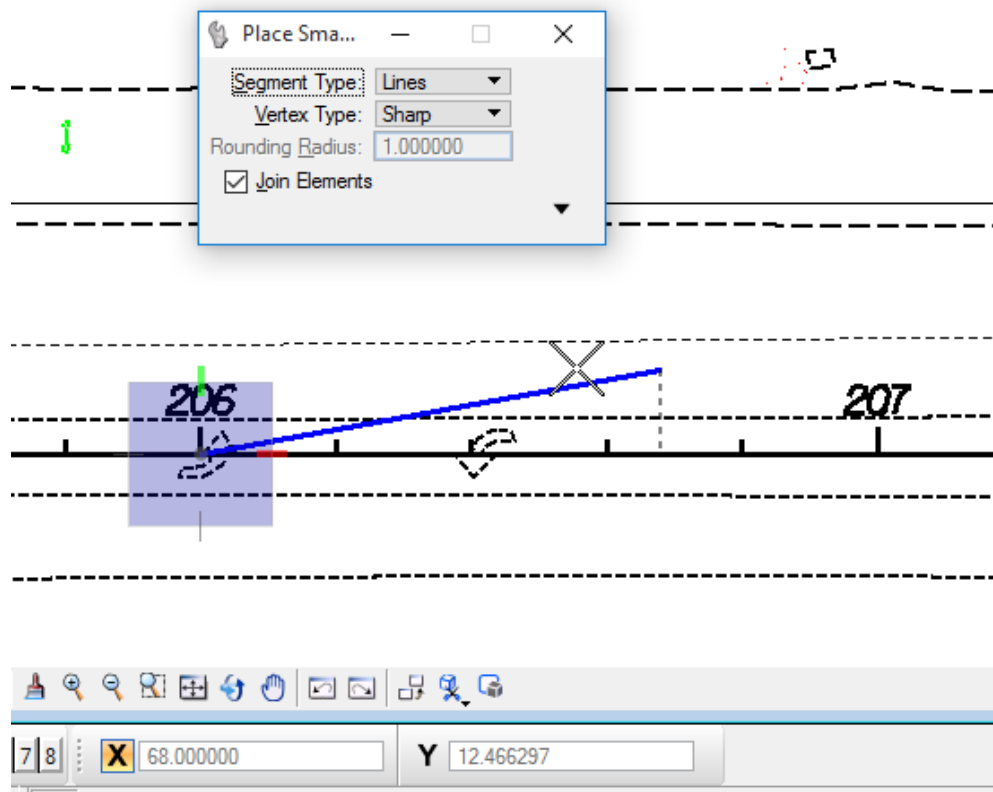
9. Move the cursor up and the Y Value in AccuDraw will highlight. Using the AccuDraw *SmartLock* feature as shown below, key-in a *distance* of **60.00.** The SmartLine extends automatically to the new location.
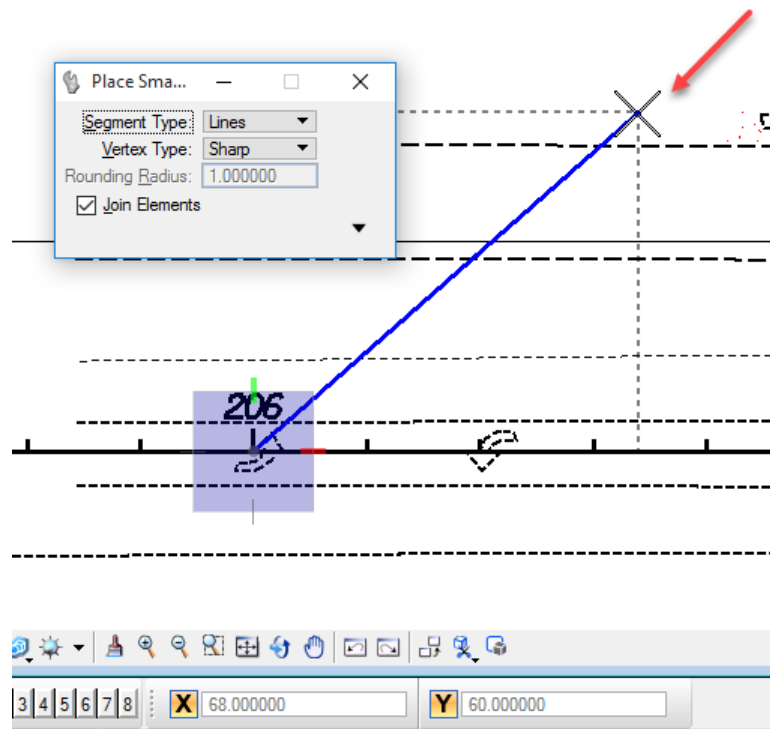
10. Data point to accept the new location of the SmartLine. Reset to exit the Place SmartLine command.

11. From the FDOT Plans Development task, click on the **Move** icon to move the fire hydrant at **Station 207+00** with a left *offset* of **70 feet** to the new location at **Station 206+68** with a left *offset* of **60 feet**.
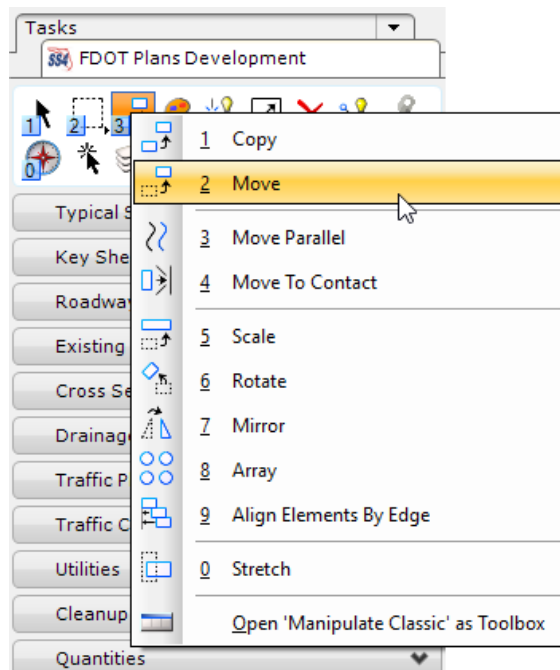
12. Snap to the center of the fire hydrant located at **Station 207+00** at a **70 foot** *offset* left of centerline and then place a Data Point to accept. The fire hydrant attaches to the cursor.
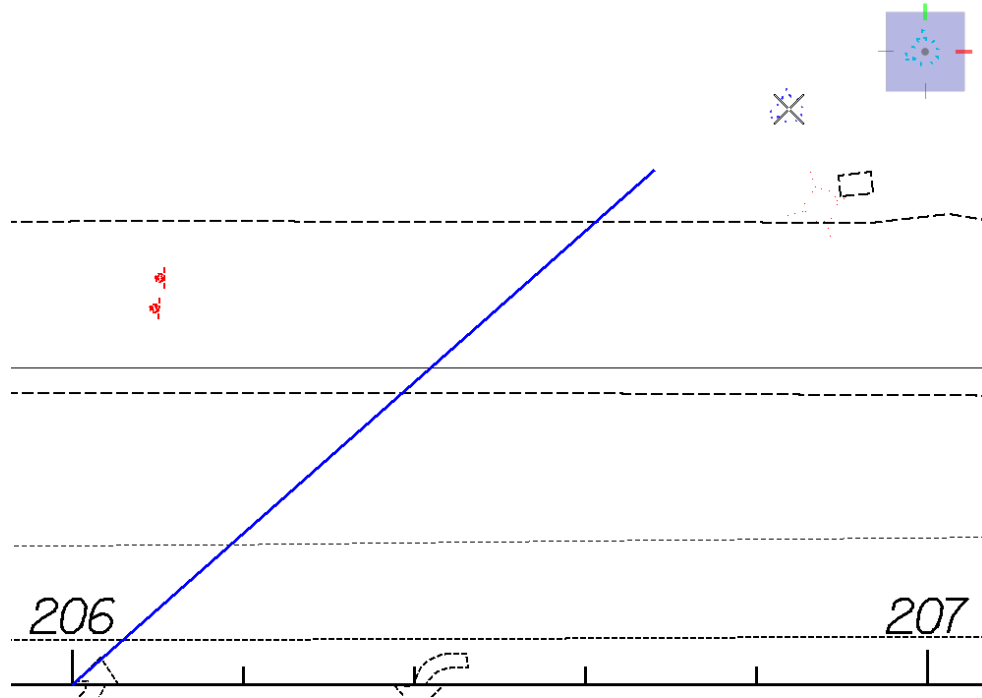
13. Snap to the new location at the end of the SmartLine previously drawn and data point to move the fire hydrant cell at the proper **Station 206+68** and *offset* **60'** left.
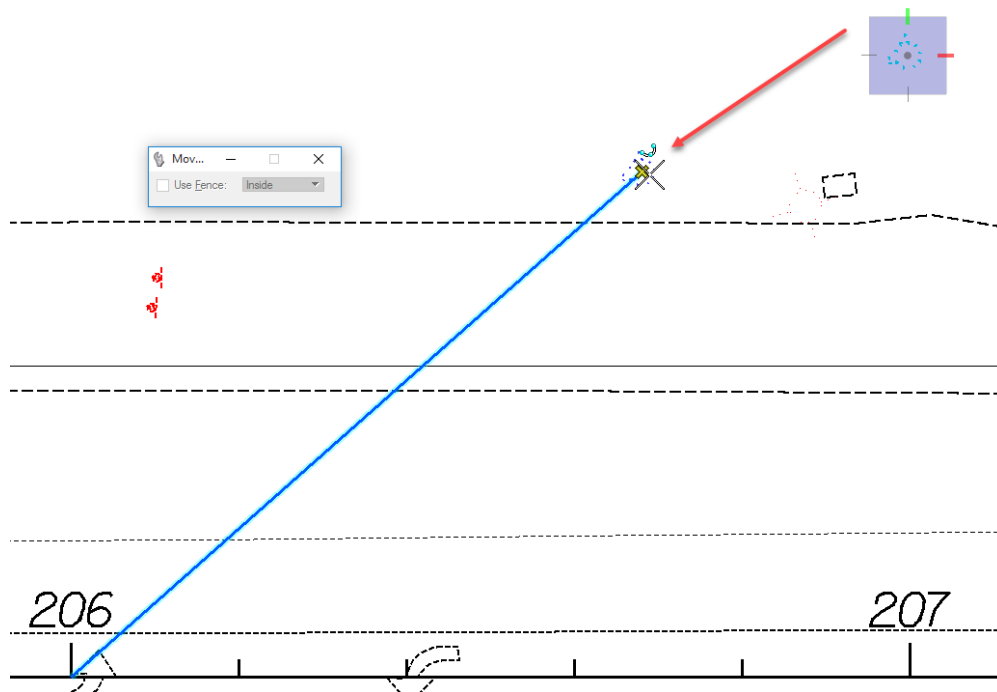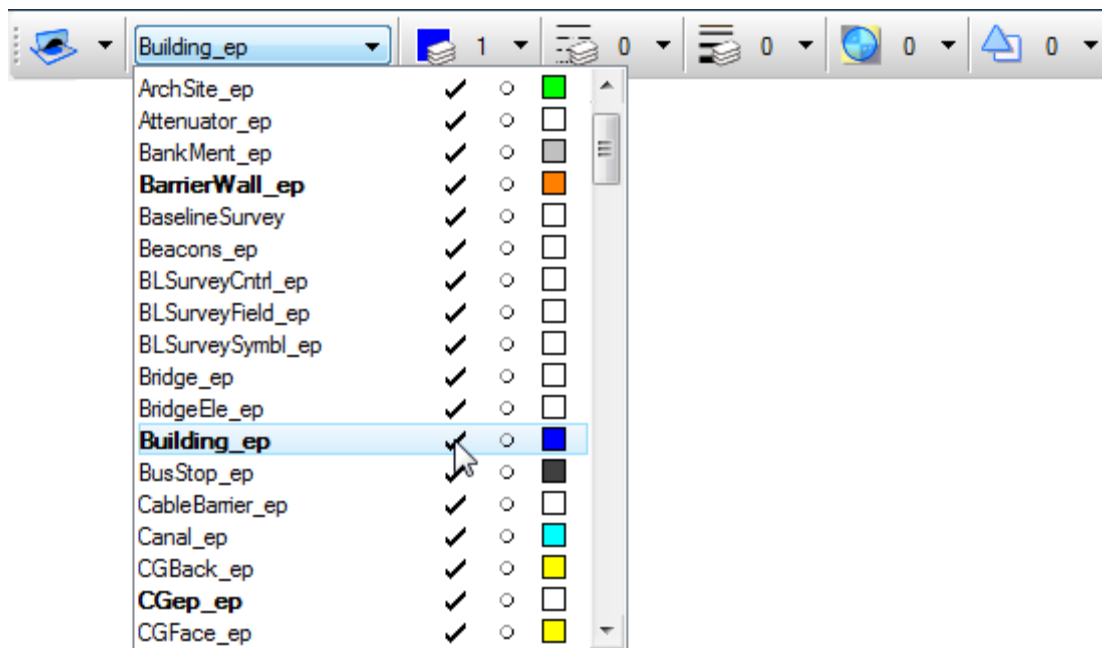
14. From the Main Task bar, click on the **Delete Element** ✕ icon and then select the SmartLine to be removed from the drawing.
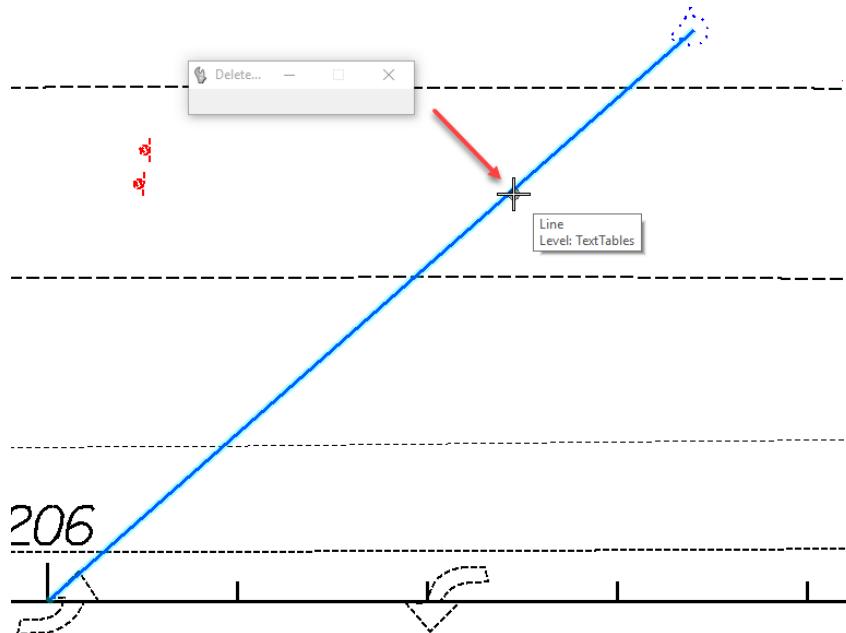

➢ *Place A Missing Building*

1. Continuing in the MicroStation File c:\e\projects\12345678901\survey*topord01.dgn*, change the *active level* to **Building_ep** from the Attributes toolbar located at the top of the application window.

2. Zoom / Pan into the location of **Stations 207+00 to 208+00**. A building not located on the original survey will be placed at this location. According to new survey field notes, the northeast corner of the building is at X=855,489, Y=1,191,686 and has the dimensions shown below.
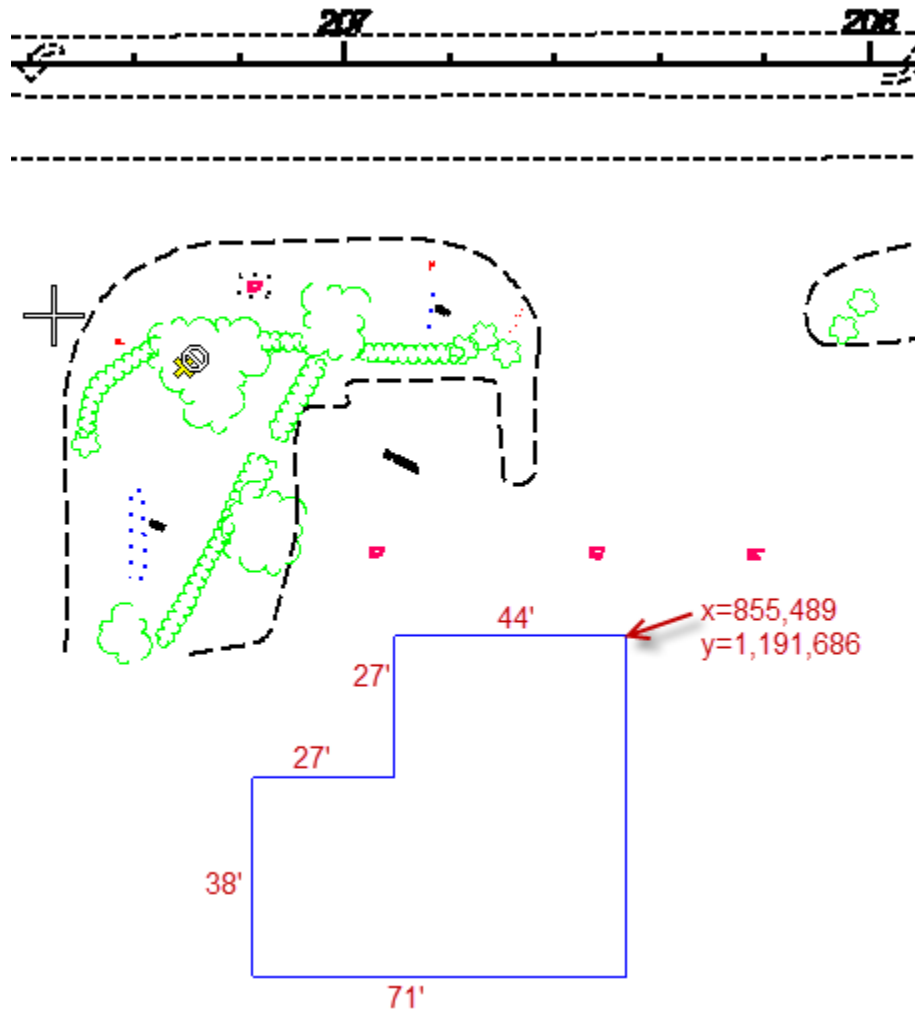
3. Ensure that AccuDraw is **active** and **undocked** into a floating dialog. If needed, click the AccuDraw dialog and press the [Space Bar] to change the AccuDraw compass to **Polar** coordinates mode as shown below.
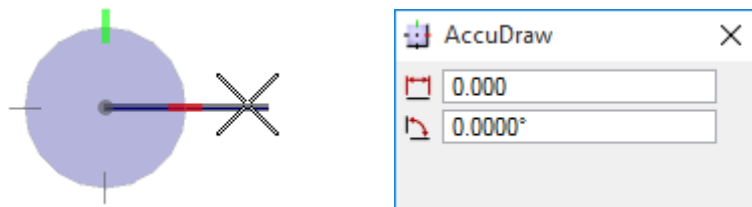
4. Click the **Place SmartLine** icon from the *Linear Elements* toolbox.

5. From the Primary tool bar, select the Key-in window and type in xy=855489,1191686, and then press the **Enter**. The SmartLine tool now begins at the northeast corner of the building.
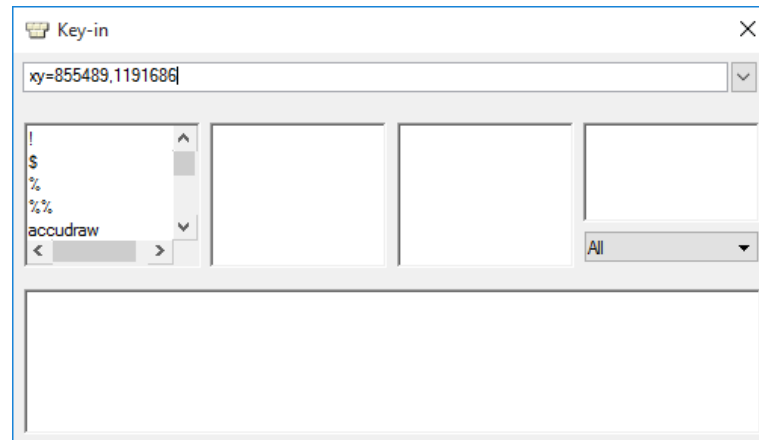

6. Move the cursor to the left to start the SmartLine. Notice the Distance highlights in the AccuDraw dialog.

7. From the AccuDraw dialog:

    a. Enter **44.00** for *Distance*, press the [Tab] key

    b. Enter **180.00** for *Angle*

    c. Place a data point. The AccuDraw Compass is now oriented to the first line segment which makes it very easy to create the building shape.
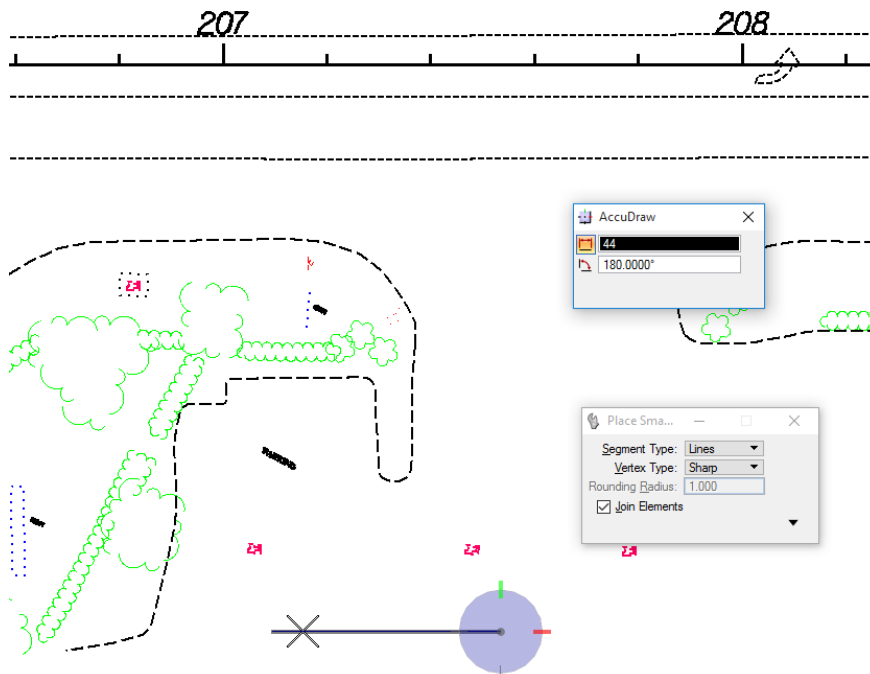
8. Complete the next side with the following:

    a. Using the AccuDraw *SmartLock* feature, move the cursor down to make the building corner **90°**.

    b. Enter a *Distance* of **27.00** in the AccuDraw dialog.

    c. Place a data point to accept the end of the line segment.
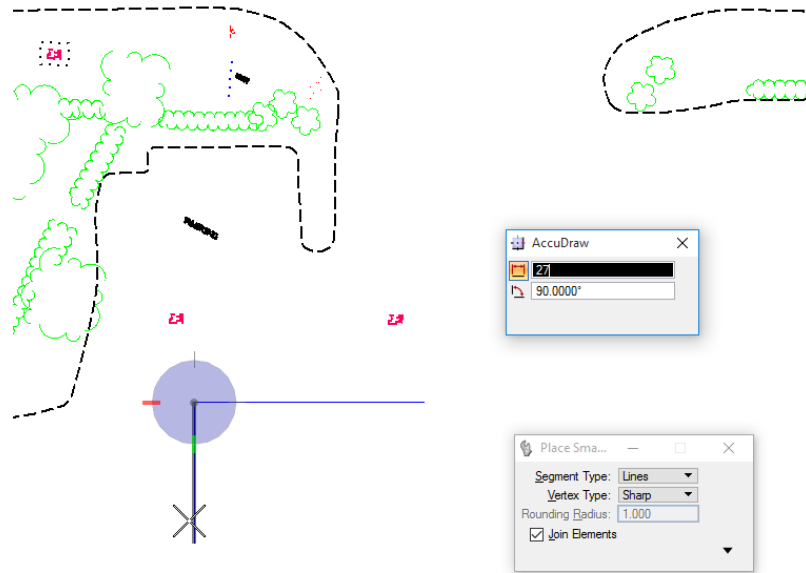
9. Repeat step 8 for the next 3 sides of the building:

    a. *Move the cursor to the left and enter Distance* of **27 feet** and make the building corner **90°**

    b. *Move the cursor down and enter Distance* of **38 feet** and make the building corner **90°**.

    c. *Move the cursor to the right and enter Distance* of **71 feet** and make the building corner **90°**.
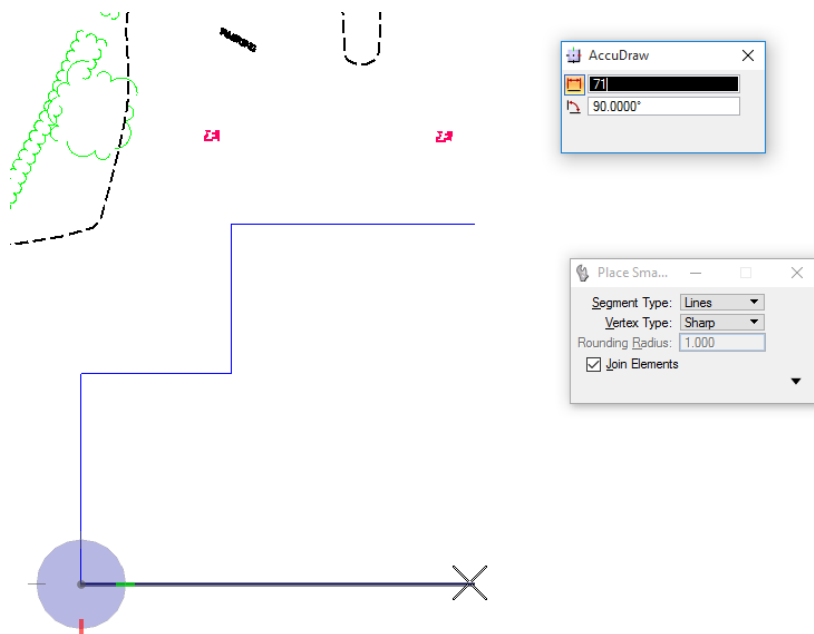
10. Move the cursor close to the starting point of the building to automatically snap to the end of the first line segment.
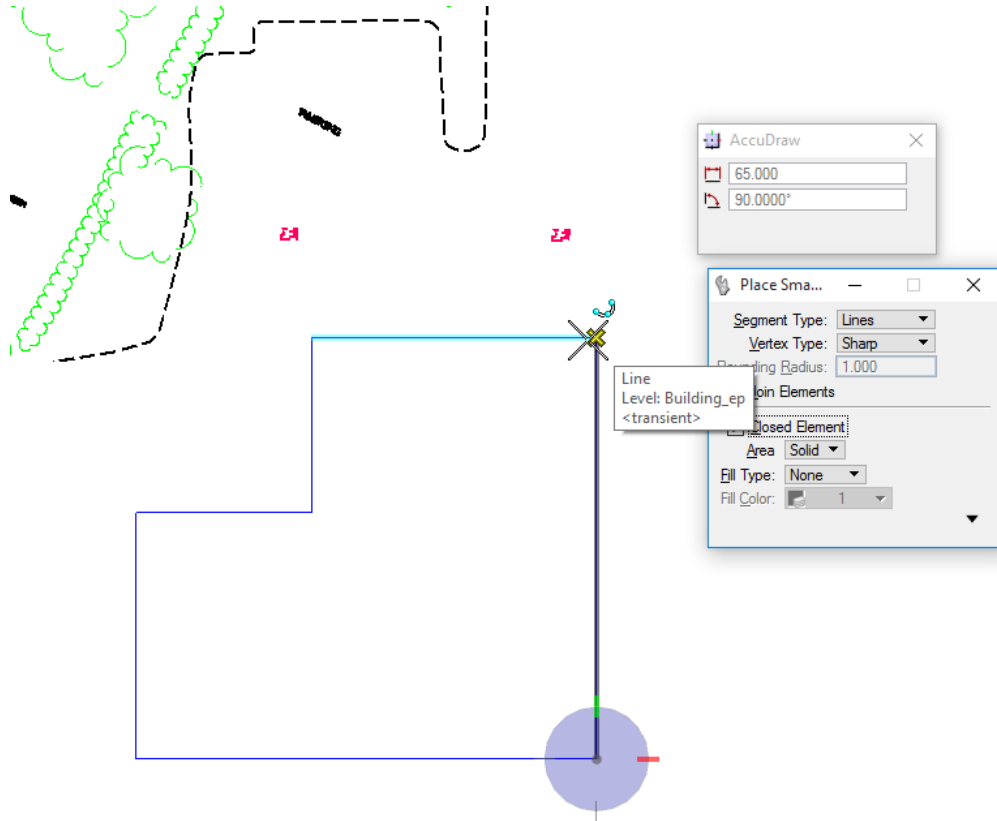
11. Place a data point to accept and then reset to end the Smartline tool.

12. When finished, **Exit** MicroStation.

8 CHANGING ELEMENTS

OBJECTIVES

Refining designs usually requires manipulating elements after they've been created. This chapter discusses the manipulation and modification tools provided in MicroStation.

- Identifying Elements

- Selecting Elements

- Manipulate Elements

- Modify Elements

- Deleting Elements

INTRODUCTION

Many times MicroStation users are not called on to create new designs, but edit existing ones. The user needs tools that allow manipulation and modification of existing geometry. MicroStation provides many tools dedicated to changing existing graphic elements.

IDENTIFYING ELEMENTS

Many of the operations performed in a design session rely on the ability to identify existing elements, or to pick specific points on those elements. MicroStation provides the tools to perform these tasks. Elements, or key points on elements, can be quickly and conveniently identified using *tentative snap points* and *data points*.
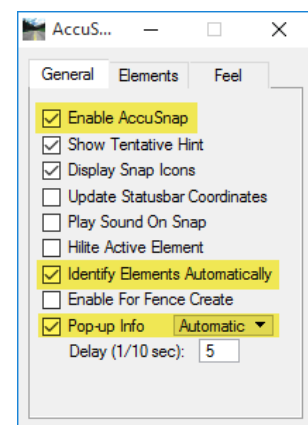
Manual Identification

Many tools require the identification of one or more elements. This can be done manually by placing the pointer over the element and entering a *data point* to highlight the element. If previewing which element is being selected, *tentative snap points* can be entered first, until the correct element highlights, and then accept with a data point. If more than one element exists in the same location, right clicking after accepting the element will scroll through the elements. Left click to accept the correct one.

AccuSnap Identification

AccuSnap automates the identification of elements and the placement of tentative snap points, so that all tentative snap points do not have to be entered manually. The *Enable AccuSnap* setting in the General tab of the AccuSnap Settings dialog can be turned on to automatically locate and snap a tentative point to elements. Simply move the cursor to the region of the element to be identified and AccuSnap snaps to it automatically.

Automatic Identification

Separately or in conjunction with AccuSnap, the capability to automatically identify elements for various tools can be enabled with the *Identify Elements Automatically* setting in the AccuSnap Settings dialog, General tab. Like AccuSnap, this setting can greatly reduce the number of button presses required in a design session. Using this setting with the Delete Element tool, for example, only requires moving the cursor over an element in any view and it highlights with a single data point to delete the element.

Popup Identification

As part of the automatic element identification functions, the *Pop-up Info* in the AccuSnap Settings dialog, General tab can be enabled. When hovering the cursor over a highlighted element, information about it appears in a pop-up window.
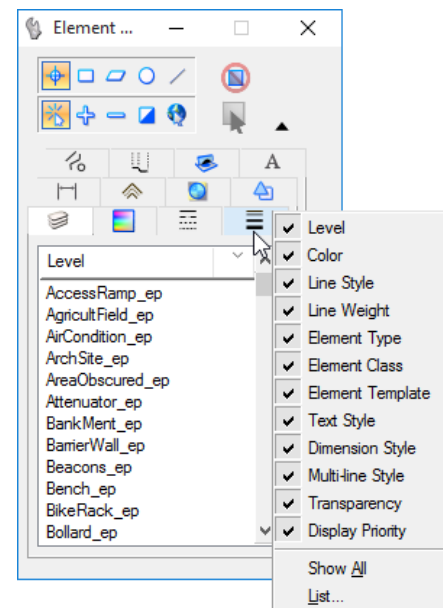
SELECTING ELEMENTS

For operations performed on existing elements within a design file, the user must have the ability to select those elements. This can be done manually, by placing the cursor over an element and entering a data point. Alternatively, the cursor can be placed over an element and enter a tentative point first, to check the selection prior to accepting with a data point. Entering a tentative point can be performed automatically with AccuSnap or by pressing the tentative button to enter it manually.

With AccuSnap's Identify Elements Automatically setting, this process is streamlined, dramatically reducing the number of button presses required. Normally, to identify an element manually requires at least one button press, or two, if entering a tentative point first. With AccuSnap and the Identify Elements Automatically setting, the cursor only has to position over an element to identify and highlight it — without a button press. For example, to delete an element with the *Delete Element* tool, the cursor need only to be placed over the element so that it highlights and then accept with a data point to complete the deletion.

Note　　The selection and grouping tools will be discussed in greater detail in Chapter 9 of this document.

Element Selection Tool

The *Element Selection* tool allows for the selection of multiple elements to be worked on as a temporary group by a variety of means as shown below. Using this method, for example, move, copy, rotate, or scale multiple elements together. By default, selected elements are bracketed with squares called *handles*. If any elements are selected, an *arrowhead* icon is displayed in the status bar. The number to the right of the icon is the number of selected elements. Chapter 9 will expand on the functionality of this tool.

Selecting By Attributes

The *Select By Attributes* dialog (Edit > Select By Attributes) provides additional attribute-based element selection options — selection by attached tag values and by properties. Properties include the area attribute (Solid or Hole), whether an element can be snapped to, whether it is locked, and whether it has been modified. Chapter 9 will expand on the functionality of this tool.
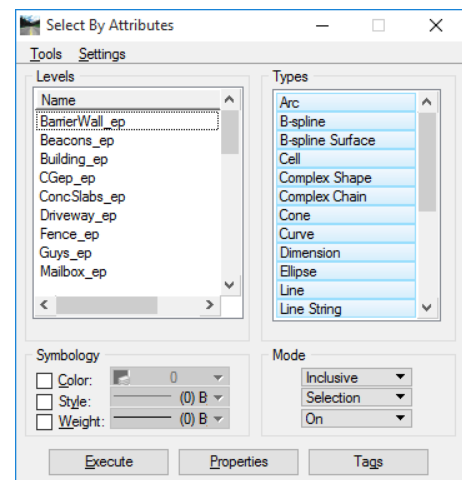
MANIPULATE ELEMENTS

The Manipulate Classic toolbox, accessed from the FDOT Plans Development task, contains tools that are used to copy, move, resize, rotate, mirror, create arrays, align, and stretch elements. These tools can be used in conjunction with the *Element Selection* tool, to manipulate groups of elements simultaneously. Of course, they can also be used by themselves to manipulate single elements.
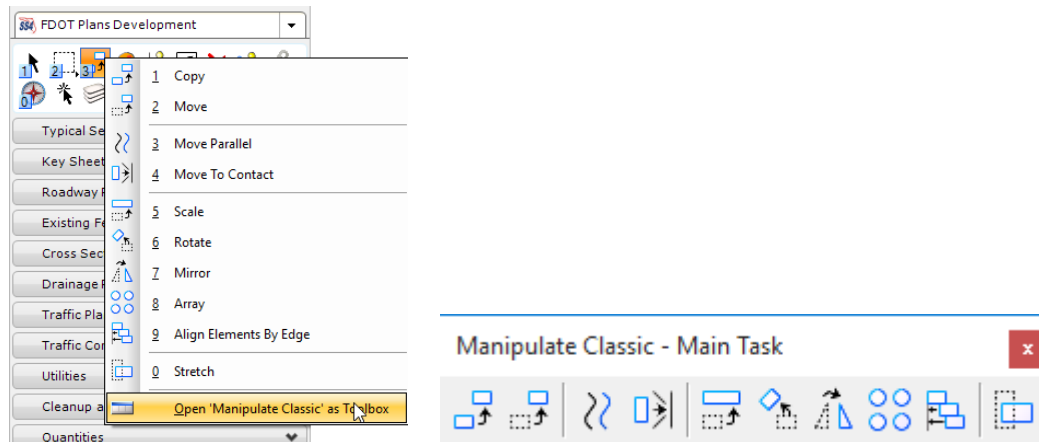

Note   Used with the Element Selection tool, an element needs only to be right-clicked on to open a menu with options to copy, move, scale, rotate, mirror, delete, or display the properties for the selected element.

	Copy Element	Copy an element(s).
	Move Element	Move an element(s).
	Move Parallel	Move or copy an element parallel to the original.
	Move to Contact	Move an element(s) in a defined direction until it contacts an existing element in the model.
	Scale Element	Resize an element(s) by the Active Scale factors.
	Rotate Element	Rotates an element(s).
	Mirror Element	Mirror an element(s).
	Array Elements	Copy an element(s) multiple times to create a rectangular or a polar (circular) array.
	Align Element by Edge	Aligns an element(s) to an edge of another element.
	Stretch Element	Stretch an element(s).

COPY ELEMENT

The *Copy Element* tool creates a duplicate of an existing element without affecting the original. It identifies the element to be copied with a data point and then enters a second data point at the location where the copy or copies should appear. An element can be copied multiple times in one operation, similar to creating an array.
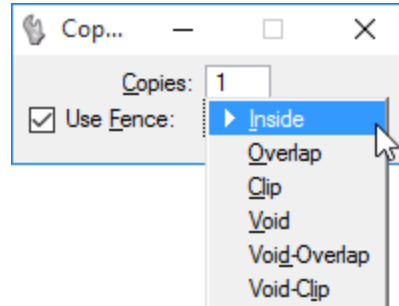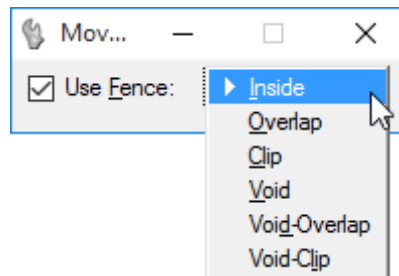

Tool settings for Copy Element are listed below:

Copies   The number of copies created is defined by the entered value.

**Use
Fence**   The fence contents are copied. This option is grayed out until an active fence is placed in the file. The fence option dropdown list sets the Fence (Selection) Mode.

MOVE ELEMENT

The **Move Element** tool relocates an element. After identifying the element to be moved, enter a second data point to define its new location. Additional data points continue to move the element until the reset command is issued.

The *Tool settings* for moving an element are described below.

Use Fence   If on, the fence contents are moved. This option is grayed out until an active fence is placed in the file. The fence option dropdown list sets the Fence (Selection) Mode.

MOVE/COPY PARALLEL

The **Move Parallel** tool is used to move or copy an element (line, line string, multi-line, curve, arc, ellipse, shape, complex chain, or complex shape) parallel to the original.
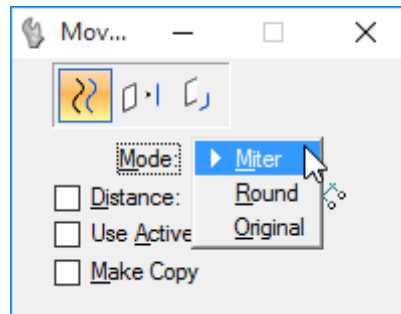
Mode	Determines the method by which the gap created, by moving two connected elements to a parallel location, is filled.
Miter	Extends or shortens connecting segments while maintaining the angle of their connections.
Round	Fills the gap with a rounded arc between the two moved elements.
Original	Fills the gap in the same manner as the MicroStation implementation of this tool; the resulting element is the same type as the original element.
Distance	If on, the entered value sets the distance to move the existing element.
Define Distance Icon	Defines the distance to move, graphically using two data points. When the distance is defined, the *Distance* setting is turned on automatically and the defined distance value is displayed in its field.
Use Active Attributes	If on, the moved or copied element takes on the active attributes. If off, the moved or copied element retains the attributes of the existing element.
Keep Original	If on, the original elements are not manipulated.

Note   For some element types, such as ellipses, Original does not produce a true parallel offset. With an ellipse Miter or Round results in a B-spline, a true parallel offset, rather than an ellipse.

MOVE TO CONTACT

The **Move To Contact** tool is used in either 2D or 3D to move one or more elements in a defined direction until they make contact with another element. If no element is in the path, then the element is not moved.
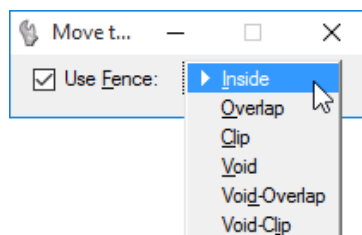
SCALE ELEMENT

The *Scale Element* tool is used to resize (scale) an element(s) individually, as a selected group, or within a fence. Scaling an element can be useful for changing the size of an element without having to redraw the element. Elements can be scaled in any or all of the three axis directions (X, Y and Z).
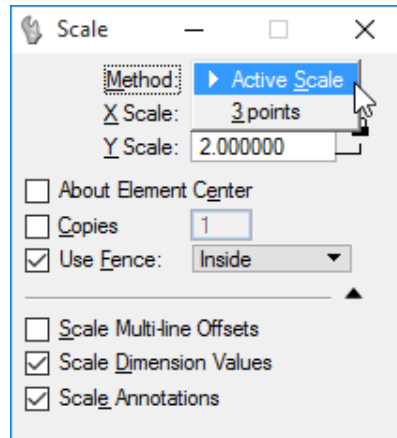
Note   Elements may be selected individually, or multiple elements selected to scale in a selection set or by using the fence.

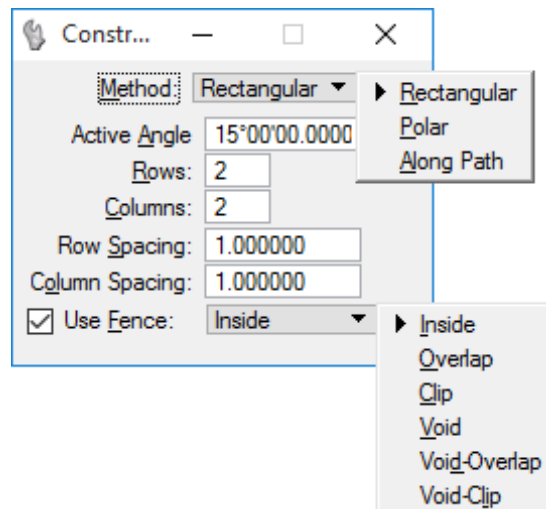| **Method**	
Active Scale	Method by which an existing element is scaled by the active scale factors (*X Scale*, *Y Scale* and *Z Scale*). If a scale factor is between 0 and 1 (for example, 0.25), the size in that direction is decreased; if a scale factor is greater than 1, the size in that direction is increased.
3 points	Method by which an existing element is scaled graphically, through the entry of three data points. The scale factors are computed by dividing the distance between the first and third points by the distance between the first and second points.
X Scale	Scale factor along view x-axis (horizontal), when method is **Active Scale**.
Y Scale	Scale factor along view y-axis (vertical), when method is **Active Scale**.
Proportional	If on, the element(s) proportions are maintained, when the **3 points** method is used.
About Element Center	If on, in the Extended Information section of the *Tool settings* dialog, the selected element(s) is scaled about its center point instead of a selected point. Cells and text elements are scaled about their origins.
Copies	If on, the element(s) are copied and the copy(s) are scaled; the original(s) are not manipulated. The adjacent field sets the number of scaled copies to create.
Use Fence	If on, the fence contents are scaled. This option is grayed out until an active fence is placed in the file and the tool is selected. The fence option menu sets the Fence (Selection) Mode.
Scale Multi-line Offsets	If on, multi-line offsets are scaled (i.e., scaling the wall thickness when resizing a room).
Scale Dimensions	If on, dimension values are modified to reflect the size of the scaled dimension. If off, only the dimension elements are scaled, while the dimension value remains unchanged.
Scale Annotations	If on, annotations are scaled. If off, size of annotations remains unchanged.

ROTATE ELEMENT

The **Rotate Element** tool is used to rotate. Elements can be rotated individually, as a selected group or within a fence.
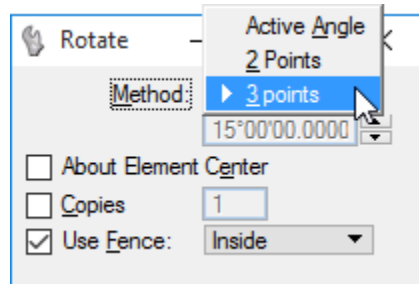
| Method	Defines how the elements are rotated.
Active Angle	Rotates elements by the Active Angle value, which can be keyed in.
2 Points	Rotates elements by an angle of rotation defined by entering two data points.
3 Points	Rotates elements by an angle of rotation defined by entering three data points.
About Element Center	If on, in the Extended Information section of the *Tool settings* dialog, the selected element(s) is rotated about its center point instead of a selected point. Cells and text elements are rotated about their origins.
Copies	If on, the element(s) are copied and the copy(s) are rotated; the original(s) are not manipulated. The adjacent field sets the number of rotated copies to create.
Use Fence	If on, the fence contents are rotated. This option is grayed out until an active fence is placed in the file. The fence option menu sets the Fence (Selection) Mode.

Note   Elements may be selected individually, or multiple elements selected to rotate in a selection set or by using the fence.

MIRROR ELEMENT

This tool mirrors an element about one of three axes. Rather than manipulating the original element, enabling the **Make Copy** toggle creates a second occurrence of the element.
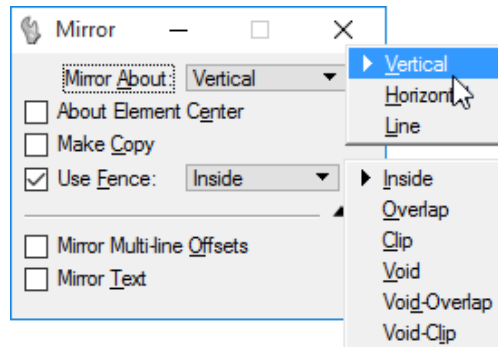
Mirror About	Sets direction in which element(s) are mirrored:
Horizontal	Elements are mirrored about the horizontal, or X, axis.
Vertical	Elements are mirrored about the about vertical, or Y, axis.
Line	Elements are mirrored about a line defined by two data points.
About Element Center	If on, the selected element(s) is mirrored about its center point instead of a selected point. Cells and text elements are mirrored about their origins.
Make Copy	If on, the element(s) are copied and the copy(s) mirrored; the original(s) are not manipulated.
Use Fence	If on, the fence contents are mirrored. This option is grayed out until an active fence is placed in the file. The fence option menu sets the Fence (Selection) Mode.
Mirror Multi-line Offsets	If on, multi-line profile offsets are mirrored.
Mirror Text	If on, text is mirrored.

CONSTRUCT ARRAY

Construct Array copies an element as many times as desired to create an array of elements. Polar arrays and rectangular arrays are supported by this tool.

Method Sets the type of array that is constructed; different *tool settings* are available depending on these settings:

Rectangular Constructs a Rectangular array.

- **Active Angle –** Aligns the imaginary orthogonal grid on which the copies are placed.

- **Rows –** Defines the number of rows in the array.

- **Columns –** Defines the number of columns in the array.

- **Row Spacing –** Defines the space between rows.

- **Column Spacing –** Defines the space between columns.

- **Use Fence –** If on, the fence contents are copied into an array. This option is grayed out until an active fence is placed in the file. The fence option menu sets the Fence (Selection) Mode.

Polar Constructs a Polar (circular) array.

- **Items –** Sets the number of items or copies in the array, including the original.

- **Delta Angle –** Sets the angle between items. If positive, copies are placed counterclockwise. If negative, copies are placed clockwise.

- **Rotate Items –** If on, the element(s) are rotated about the center of the array.

- **Use Fence –** If on, the fence contents are copied into an array. This option is grayed out until an active fence is placed in the file. The fence option menu sets the Fence (Selection) Mode.

Along Path Constructs an array along a path element.

- **Mode –** Sets the way that the array is created.

  - *Number* – Defines the number of items in the array, via the Copies setting.

  - *Distance* – Defines the distance between items in the array, via the Distance setting.

  - *Both* – Defines the number of items in the array and the distance between each item, via the Copies and Distance settings respectively.

- **Copies –** If on, the element(s) are copied; the original(s) are not manipulated. The adjacent field sets the number of rotated copies to create.

- **Rotate Items –** If on, the element(s) are rotated to maintain their relative orientation to the path element.

- **Use Fence –** If on, the fence contents are copied into an array. This option is grayed out until an active fence is placed in the file. The fence option menu sets the Fence (Selection) Mode.
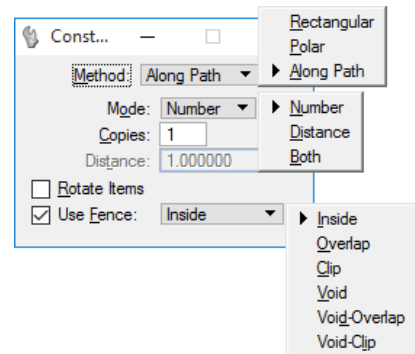
ALIGN ELEMENTS BY EDGE

This tool is used to align an element or group of elements to the edge of a secondary element.
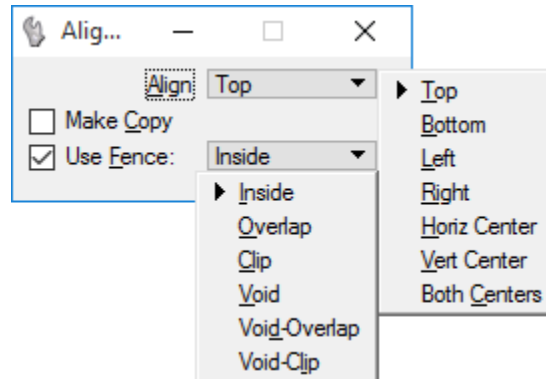
Align	Sets the manner by which to align.
Make Copy	If on, the element(s) are copied and the copy(s) are aligned to the identified base element; the original(s) are not manipulated.
Use Fence	If on, the fence contents are aligned. This option is grayed out until an active fence is placed in the file. The option menu sets the Fence (Selection) Mode.

STRETCH ELEMENT

The **Stretch Element** tool is used to manipulate breaks (such as breaks in multi-lines, which are often used to represent windows or doorways when the multi-line itself is the wall).

The **Stretch Element** tool operates on elements as follows:

- A vertex of a line, line string, multi-line, or shape is moved if it is inside the fence. A vertex outside the fence is not moved.

- An endpoint of an arc is moved if it is inside the fence. An arc endpoint outside the fence is not moved.

- An ellipse, circle, or cell is moved if it is completely inside the fence.

- One or more poles (control points) of a B-spline surface, that are located inside the fence, are moved.
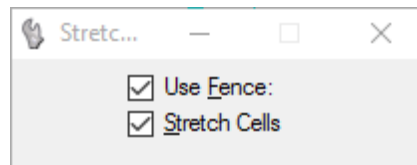
Use Fence	(Fence present only) If on, the current active fence is used.
Stretch Cells	If on, cells that overlap the fence are stretched.

Exercise 8.1 *Manipulate Tools*

➢ *Copy Parallel*

Construct an edge of pavement line.

1. Open the MicroStation file c:\e\projects\12345678901\survey*topord02.dgn*.

2. Pan and Zoom into the area of the **7-11 Convenience Store** located on the East side of the CL Survey near **Station 313+00**.
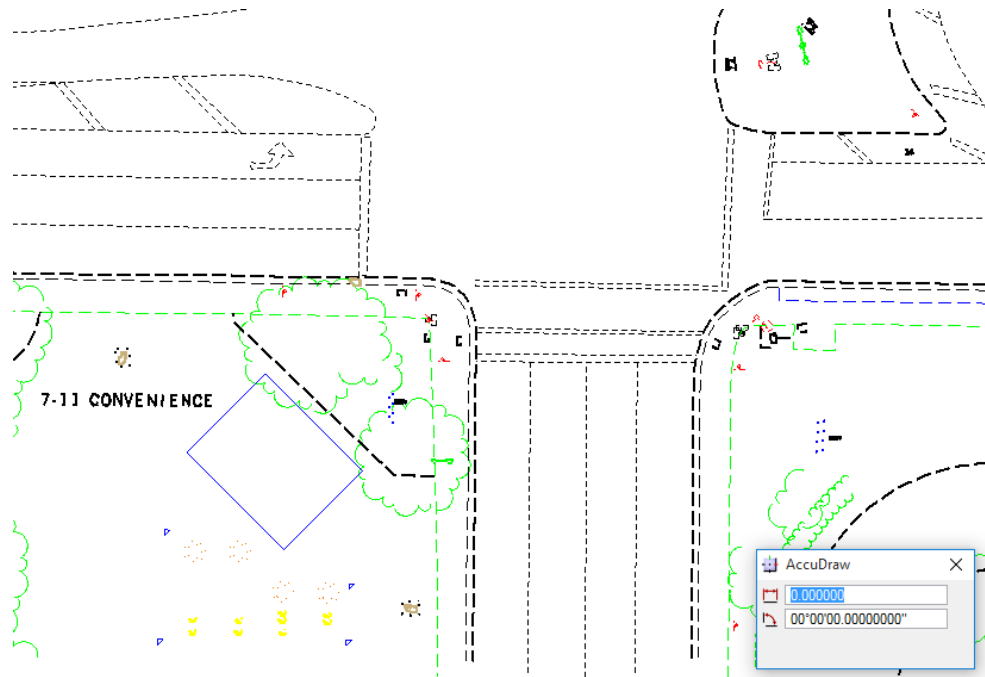
3. From the *Manipulate* toolbox, click the **Move Parallel** icon.
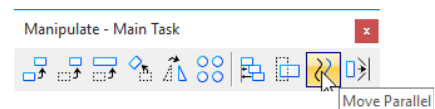
4. In the Move/Copy Parallel Tool Settings, enter the values shown to construct an *edge of pavement line* **1.5 feet** inside the field located at the back of curb line that's shown around the northern edge of the parking area.
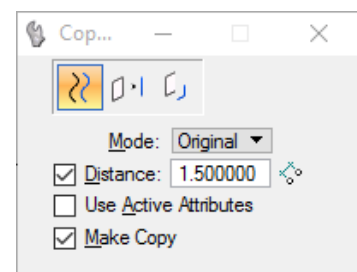
Tip *Remember to watch the step-by-step reminder messages located in the lower left side of the Status Bar.*

5. Select the **topo miscellaneous line**, as shown below.
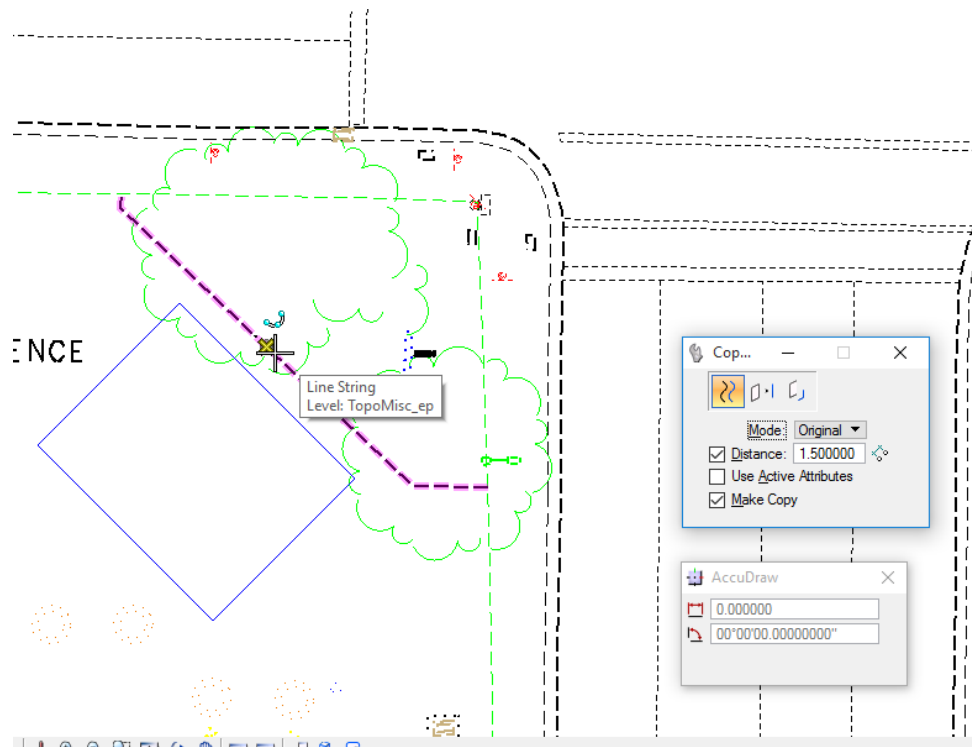
6. Move the cursor to the right. A copy will be automatically display **1.5 feet** from the back of curb line. Place data point to accept as highlighted below.
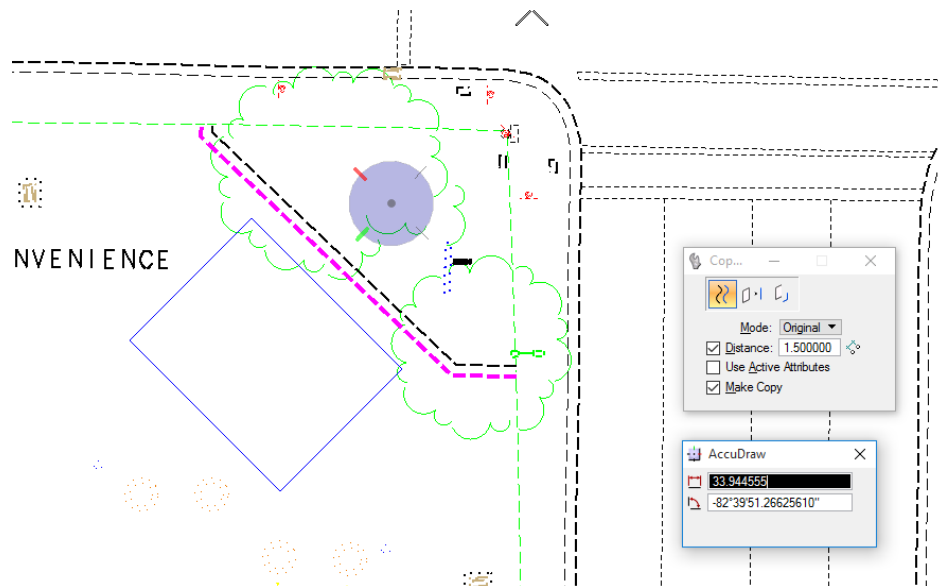
7. Right-click to reset and close the *Copy Parallel* tool.

➢ *Rotate Element*

1. Continuing in c:\e\projects\12345678901\survey*topord02.dgn,* zoom into the **sign** located on the north side of the 7-11 lot.

2. From the *Manipulate* toolbox, click the **Rotate** icon.
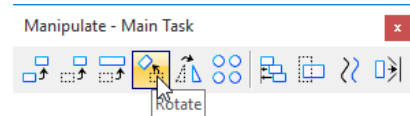
3. To rotate the sign to be parallel with the curb, enter the values shown in the *Rotate* Tool Settings.
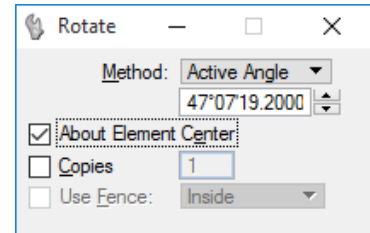
4. Select the **sign** element. The image automatically displays the rotation.
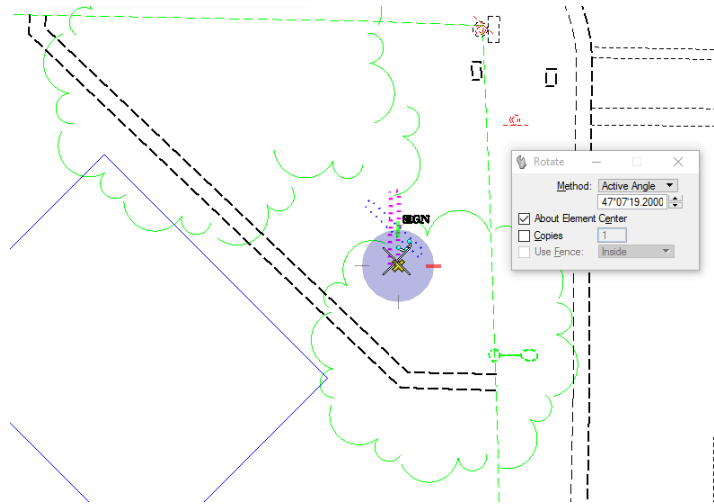
5. Place a data point anywhere to accept the rotation.
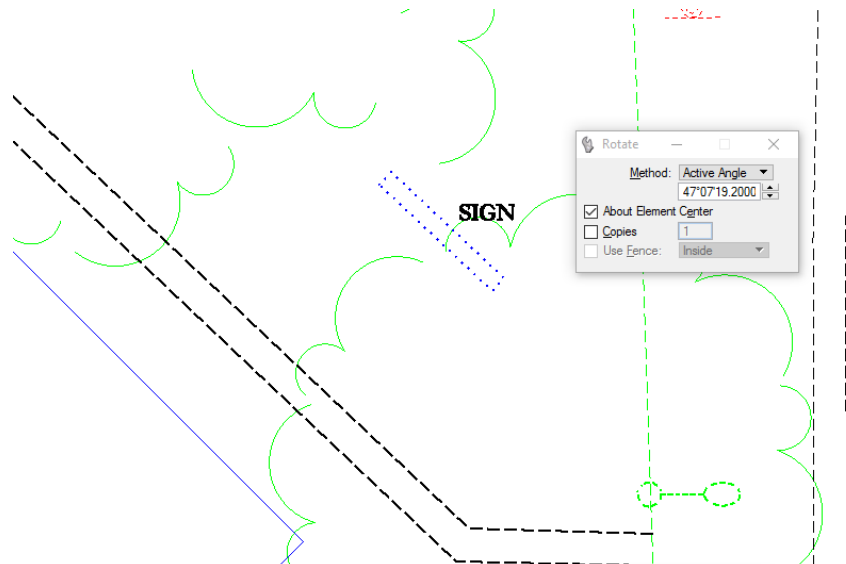
> ➢ *Align Edges*

1. Continuing on the 7-11 lot, using the *View Controls*, window-in on the area of the gas pumps as shown below.
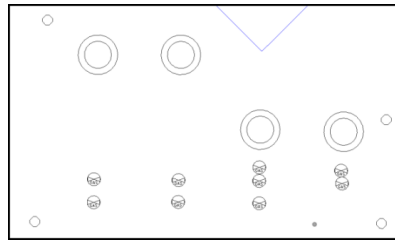
2. Click on the **Align Edges** icon located on the *Manipulate* toolbox.
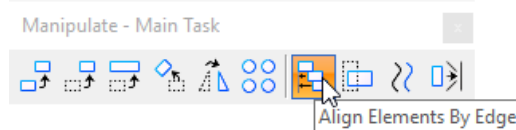
3. Match the *Align Edges* Tool Settings shown below.
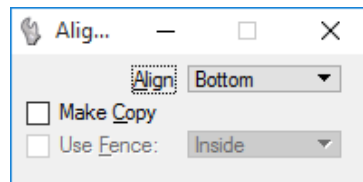
4. Click on the bottom of **circle number 1** as the *base element for alignment*.
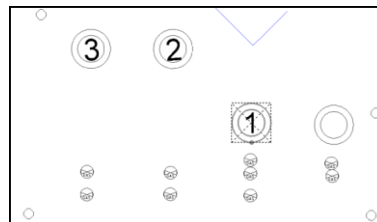
5. Click on the bottom of **circles number 2** and **3** to align with *circle number 1*.
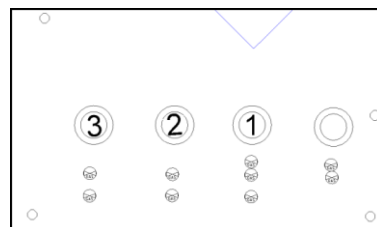
Tip   MicroStation *provides next step reminders in the lower left status bar.*

 then

6. Place a **data point** to accept selections and close the *Align Edges* tool.

MODIFY ELEMENTS

The Modify Element toolbox, accessed from the Main toolbox (or from the Task Navigator by selecting **Main > Modify**), contains tools that are used to modify the geometry of an element; usually by adding or deleting geometry or vertex points.
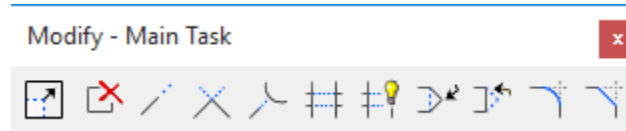

	Modify Element	One of the most useful tools as it can be used to modify many types of geometry.
	Partial Delete	Delete part of an element.
	Break Element by Point	Break a linear element at a defined point.
	Extend Line	Extend or shorten a line.
	Trim to Intersection	Extend or shorten two open elements to their intersection.
	Trim to Element	Extend or shorten an open element to its intersection with another element.
	Trim Multiple	Trim, extend, or trim and extend elements one at a time, or multiple elements simultaneously, to their intersection with one or more other elements.
	IntelliTrim	Simultaneously trim or extend multiple elements at their intersection with one or more other elements.
	Insert Vertex	Insert a vertex in a linear element.
	Delete Vertex	Delete a vertex from a linear element.
	Construct Circular Fillet	Construct a circular fillet between two elements.
	Construct Chamfer	Construct a chamfer between two lines or adjacent segments of a line string or shape.

MODIFY ELEMENT

The *Modify Element* tool is used most often to adjust lines and line segments by moving or inserting vertices. It can also be used to change the geometry of blocks and complex shapes. This tool is used to do the following:

- Move a vertex or segment of a line, line string, multi-line, curve, B-spline control polygon, shape, complex chain, or complex shape.
- Scale a block about the opposite vertex.
- Modify rounded segments of complex chains and complex shapes created with the *Place SmartLine* tool while preserving their tangency.
- Change rounded segments of complex chains and complex shapes to sharp and vice-versa.
- Scale a circular arc while maintaining its sweep angle.
- Change a circle's radius or the length of one axis of an ellipse.
- Move dimension text or modify the extension line length of a dimension element.

Elements with multiple segments are line strings, shapes, and complex chains, including complex shapes placed with the **Place SmartLine** tool. When used with AccuDraw, the *Modify Element* tool not only modifies vertices and segments, but also changes a vertex's type (*Sharp, Rounded or Chamfered*).
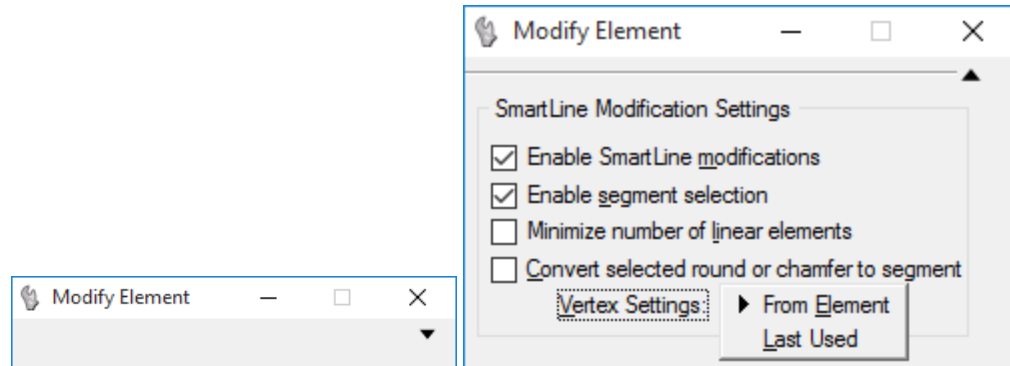
SmartLine Modification Settings	Displays the controls for defining how SmartLine elements are modified with the *Modify Element* tool.
	Turning off this setting disables the remaining settings as they apply only to SmartLine style modifications.
Enable SmartLine modifications	If on (the default), modifications to vertices and segments of SmartLines take into account the adjoining elements. For example, lines that are tangential to a rounded vertex, remain tangential.
	If off, vertices and segments of SmartLines are modified individually.
Enable segment selection	If on (the default), segments of SmartLines may be selected, as well as the vertices. If off, only vertices may be modified.
Minimize number of linear elements	If on, attempts to reduce the number of component elements in a complex chain or shape. For example, a number of individual lines that have been chained together may be converted to a line string.
	If off, does not attempt to reduce the number of component elements in a complex chain or shape.
Convert selected round or chamfer to segment	If on, when a rounded or chamfered vertex is selected, it is converted to a segment (arc or linear segment) so that it can be modified as such. This applies only to the selected round or chamfer, not to the whole chain or shape. After modification, the component will not be treated as a round or chamfer again.
Vertex Settings	Defines the type of vertex to use.

- *From Element* — Uses the element's own settings.

- *Last Used* — Uses the type of vertex that was last used when creating a SmartLine element.

PARTIAL DELETE

The *Partial Delete* tool is used to delete parts of, but not the entire element. The first data point defines the start point of the deletion and the second data point defines the direction and extent of the deletion. Two rules are followed when used:

- A closed element is turned into an open element (i.e., an ellipse or circle becomes an arc, a shape becomes a line string, a closed B-spline curve becomes an open B-spline curve, etc.).

- If an interior portion of an open element (line, line string, multi-line, curve, or arc) is partially deleted, the element is divided into two elements of the same type.

BREAK ELEMENT

The *Break Element* tool is used to break a linear element at a defined point by identifying the element at the point where the break is required with a data point and then accepting to place the break.
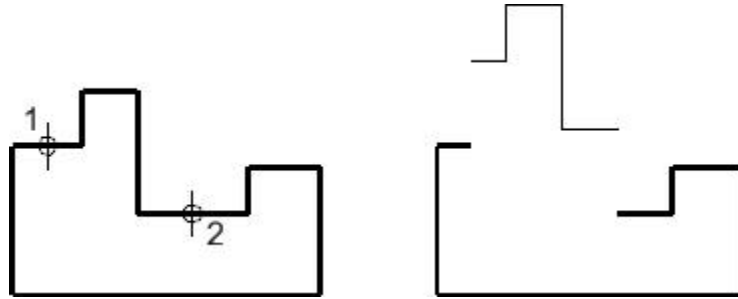
Right: After placing break points, the two sections can be manipulated individually.

EXTEND LINE

The **Extend Line** tool is used to extend or shorten a line or an end segment of a line string or multi-line.
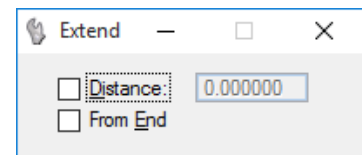
	Distance	Value

Distance   If on, the entered value specifies the distance to extend or shorten the line.

- A negative distance shortens the line.
- A positive distance extends the line

From End   *(Applies for Distance off only)*   If on, the extension, or shortening, of the line is relative to the nearest endpoint to the identification point.

If off, the extension, or shortening, of the line is relative to the origin point of the line, no matter where it is identified.

TRIM TO INTERSECTION

The *Trim To Intersection* tool is used to extend or shorten two elements of any type to their intersection. When an element is extended, the end nearest the point where it was selected is moved, and the other end remains unchanged. When an element is shortened, the part of the element that is selected is kept.
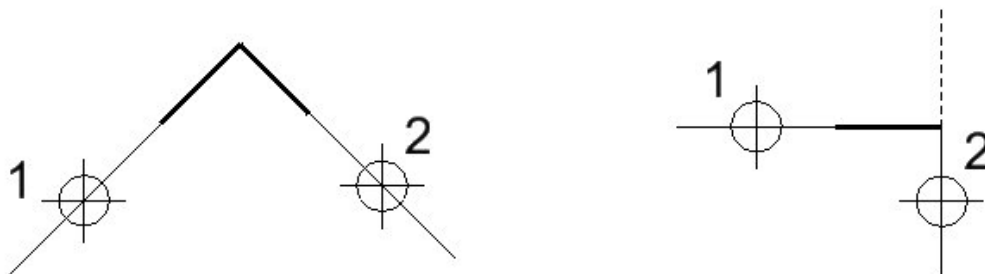
TRIM TO ELEMENT

The *Trim to Element* tool is used to extend or shorten one or more lines, line strings, or arcs to their intersection with another element. When an element is extended, the end nearest the point where it was identified is moved, and the other end remains unchanged. When an element is shortened, the part of the element that is identified is kept.
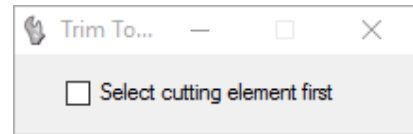
Select cutting element first	If on, the element selected first will be used as the cutting element and the element(s) selected second will be the element that is extended or shortened. The default is off.
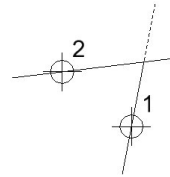
TRIM MULTIPLE

The *Trim Element* tool is used to trim or cut an element or series of elements at their intersection with one or more cutting elements. The cutting element and the element that is trimmed can be lines, line strings, arcs, curves, B-spline curves, shapes, ellipses, complex chains or complex shapes.
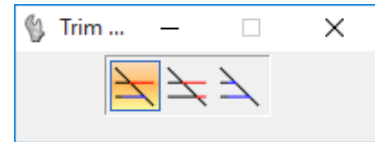
Trim	If selected, the elements that intersect with the cutting element(s) will be trimmed.
Trim and Extend	If selected, the elements that intersect with the cutting element(s) will be trimmed, and the selected elements that can be extended to intersect with the cutting element(s) will be extended.
Extend	If selected, the elements that can be extended to intersect with the cutting element(s) will be extended.

INTELLITRIM

The *IntilliTrim* tool is used to simultaneously trim, extend or cut multiple elements. The cutting elements or elements which can extend other elements are lines, line strings, ellipses, arcs, curves, shapes, complex strings, complex shapes, text nodes, and cell headers. Elements that can be cut or trimmed are lines, line strings, curves, arcs, ellipses, shapes, B-spline curves, complex strings, and complex shapes. The only elements that can be extended are lines, line strings, complex chains that end with a line or line string, and b-spline curves.
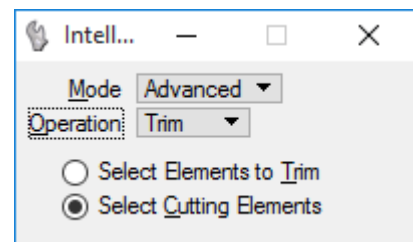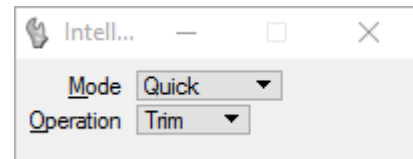
|	
Quick Mode	• *Trim* — If on, elements are trimmed.
	• *Extend*— If on, elements are extended.
	• *Cut* — If on, elements are cut. (Where the element was cut will not be seen until it is selected.)
Advanced Mode	• *Trim* — If on, elements are trimmed.
	• *Extend* — If on, elements are extended.
	• *Cut* — Dimmed if Advanced is selected.

Select Elements to Trim If selected, each element that is identified is designated as an element to trim (or extend)

Select Cutting Elements If selected, each element that is identified is designated as a cutting element.

INSERT VERTEX

The *Insert Vertex* tool inserts additional vertices within a line, line string, multi-line, shape, or B-spline control polygon by identifying the element, then entering a data point to position the new vertex.
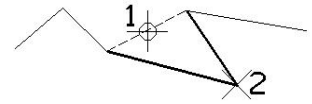
This tool can be used to attach a line segment to the endpoint of a line or line string, add an extension line to a dimension element, and extend a point curve. It can also be used to change a point element into a line or a line element into a line string

Note   A shape or line string can have 5000 vertices.

DELETE VERTEX

The *Delete Vertex* tool works very much like the **Insert Vertex** tool, except in reverse. It is used to remove vertices from line strings, multi-line, shape, or B-spline control polygon by identifying the vertex or extension line and accepting the deletion. It is also used to delete an extension line from a dimension element.
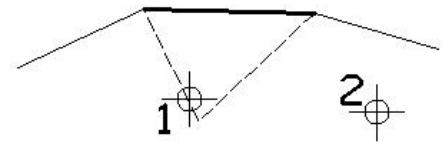
CONSTRUCT CIRCULAR FILLET

The *Construct Circular Fillet* tool is used to construct a circular fillet (arc) between two elements (lines, line strings, circular arcs, circles, or shapes), two segments of a line string, or two sides of a shape.
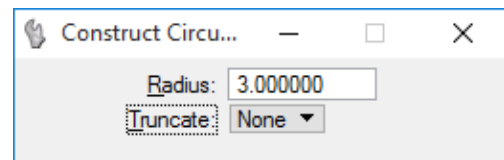
Radius   Sets the fillet's radius.

Truncate   Sets which side(s) are truncated.

   None — Neither element or segment is truncated.

   Both — Both elements or segments are truncated at their point of tangency with the fillet.

   First — The first element or segment identified (step 2) is truncated at its point of tangency with the fillet.
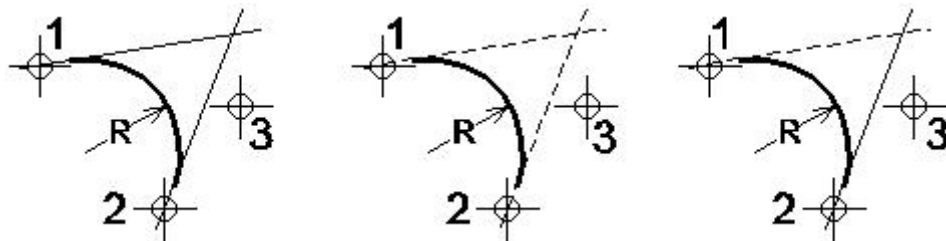
Construct Circular Fillet. From left: Truncate set to None, Both, and First

CONSTRUCT CHAMFER

The *Construct Chamfer* tool is used to construct a chamfer between either of the following:

- Two lines — The original lines are trimmed and a third line element forms the chamfer.

- Adjacent segments of a line string or shape — An additional vertex is inserted and the common vertex is adjusted to form the chamfer.
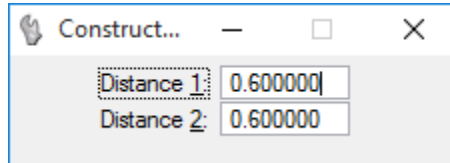
| Distance **1** | Applies to the first element or segment identified. (Step 2) |
| Distance **2** | Applies to the second element or segment identified. (Step 3) |
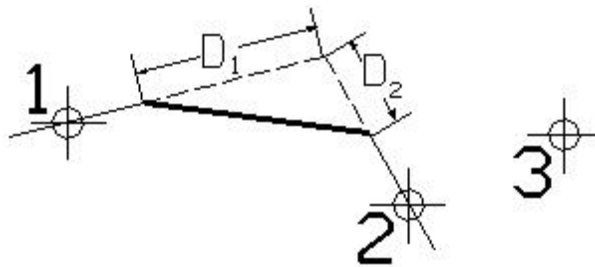
Chamfer D1 denotes Distance 1. D2 denotes Distance 2

Exercise 8.2 Modify Tools

➢ **Extend Line**

1. Continuing in c:\e\projects\12345678901\survey*topord02.dgn*, zoom into **Stations 28+00 and 29+00**, just across the street from the 7-11 Convenience.
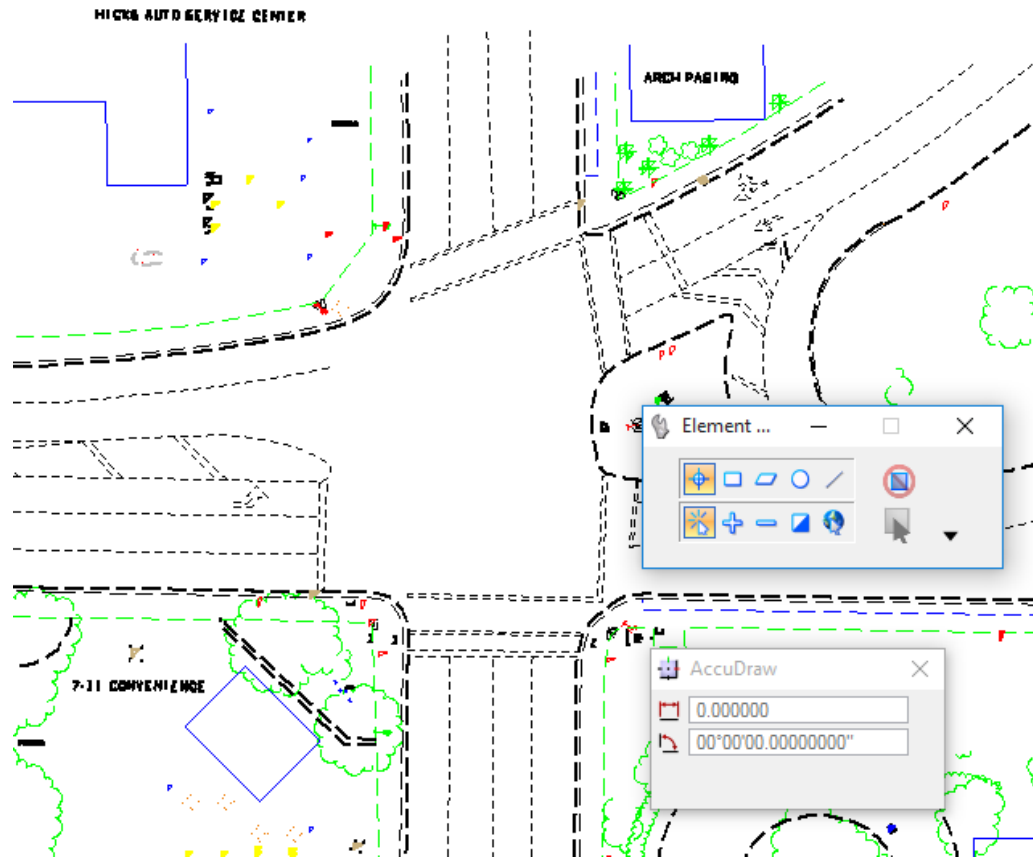
2. From the *Primary* toolbox located at the top of the application window, confirm that AccuDraw is **active** by toggling the ⊞ icon. There will be a floating dialog or it may be docked at the bottom of the application window.
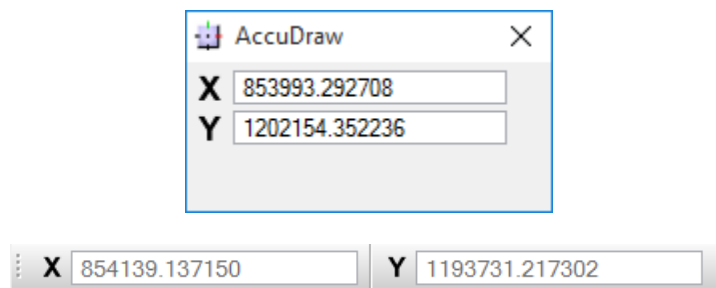
3. From the Main task, under the *Modify* toolbox, click the **Extend Line** icon and leave the *Extend* Tool Settings as shown.
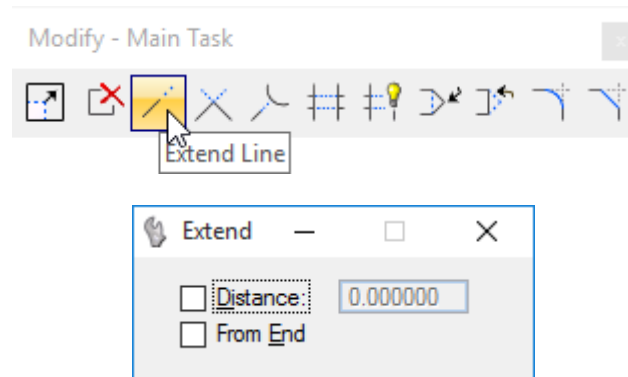
Tip   *Watch for the* MicroStation *reminder messages in the lower left side of the Status Bar.*

4. Using the tick mark at **Station 28+00** as the *limit for extension*, extend all pavement, lane, curb and sidewalk lines to that limit:

    a. Move the cursor near one of the elements. There should be an automatic snap to the element. Place a data point to accept.
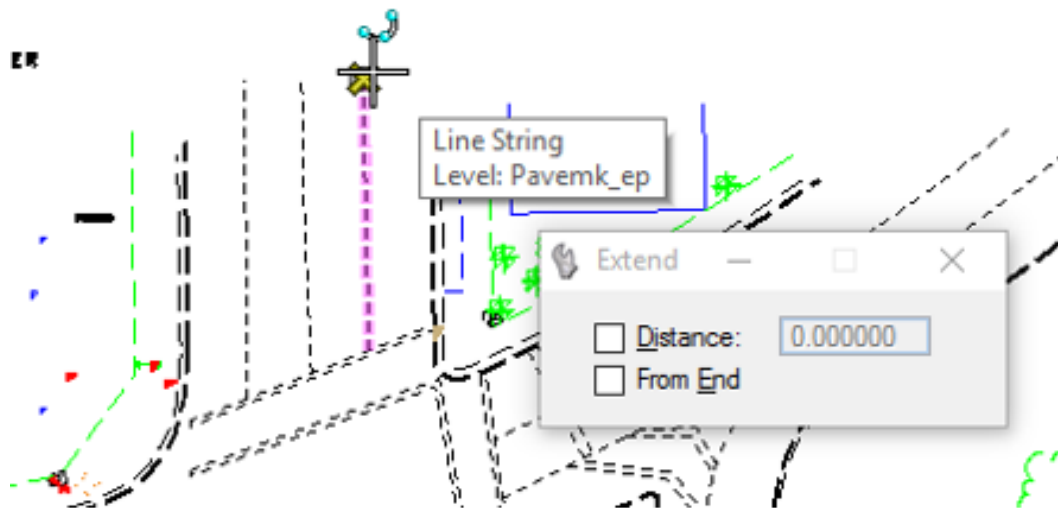
b. Enter **20** into the X value of AccuDraw, extending the line upwards. Place a data point to accept.
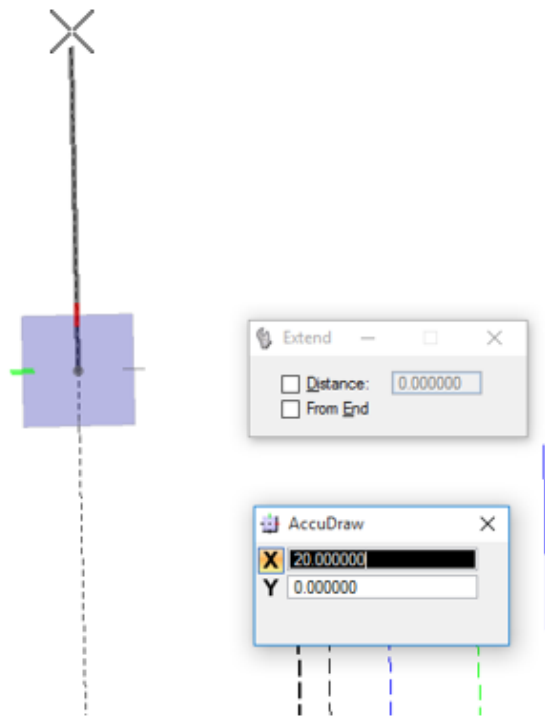
5. Repeat that process, striking ENTER to lock the X Axis in AccuDraw and snapping to the previous line until the drawing looks like the survey below.
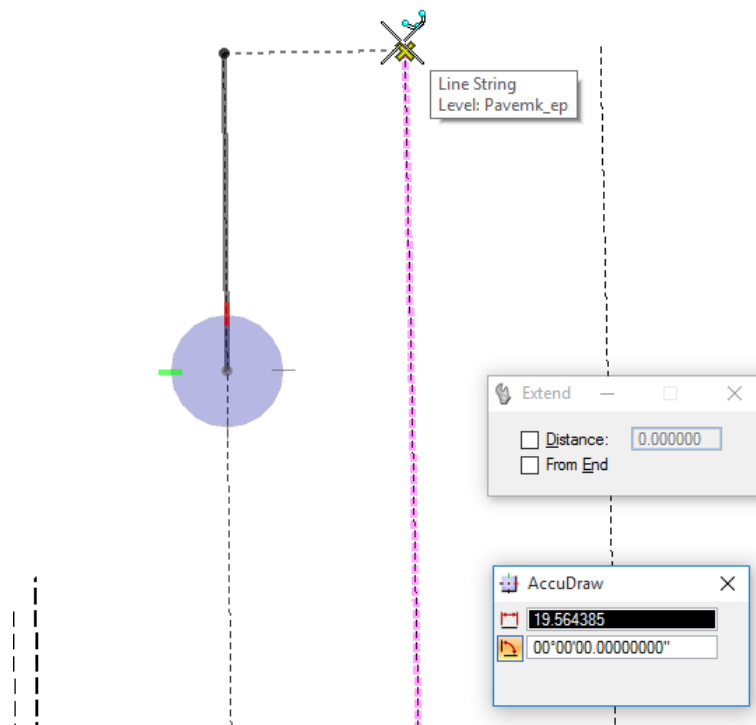
DELETE ELEMENTS

Up to this point, all of the tools covered have been geometry creation tools. Deleting geometry is also very important. The *Delete Element* tool is one of the most used tools in the Main toolbox. It is used to remove an element(s) from the design by either of the following:

- Select an element or group of elements to delete with the Element Selection tool and then select Delete Element.

  <OR>

- Select Delete Element first and then enter a data point on each of the elements to be deleted.
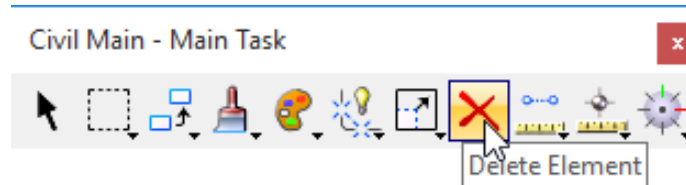
Tip   *With the Element Selection tool method, the Delete key on the keyboard can be used instead of selecting Delete Element.*

AccuSnap, discussed in Chapter 7, is an intelligent element locating feature of MicroStation and can be used to identify elements automatically for deletion if the *Identify Elements Automatically* checkbox is populated on the AccuSnap Settings dialog.
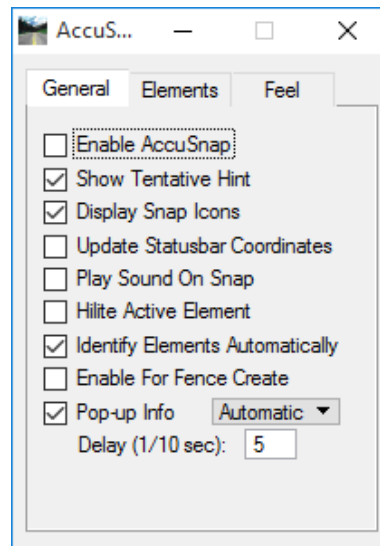
The *Delete Fence Contents* tool in the Fence toolbox, discussed in Chapter 9, can also be used to delete a group of elements.
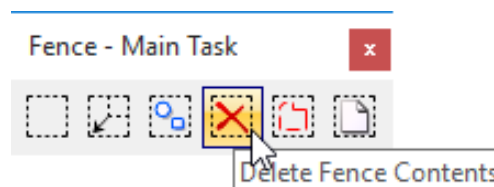
9 SELECTING & GROUPING

OBJECTIVES

MicroStation has several tools that allow multiple elements to be selected and grouped together. These include:

- Element Selection Tool
- Fences Tools
- Select by Attributes
- Grouping Elements

INTRODUCTION

To this point, any modifications and manipulations have been performed on single elements. When working with actual drawings, this is often not practical. A more common task is to modify or manipulate a group of elements. For instance, all direction arrows may need to be erased, or an area of the drawing may need to be moved or copied to a new location.

MicroStation has several tools that allow multiple elements to be selected. The next several topics cover various ways of selecting multiple objects in MicroStation.

ELEMENT SELECTION TOOL

For operations on existing elements in the DGN file, they must be selectable. This can be done manually, by placing the cursor over the element and entering a data point. Alternatively, place the cursor over the element and enter a tentative point first, to check the selection prior to accepting with a data point. Entering a tentative point can be performed automatically with AccuSnap or by pressing the tentative button to enter it manually.

For operations on several elements, simultaneously, use the *Element Selection* tool. This tool allows selection of multiple elements to be worked on as a temporary group. Using this method, for example, move, copy, rotate, or scale multiple elements together.
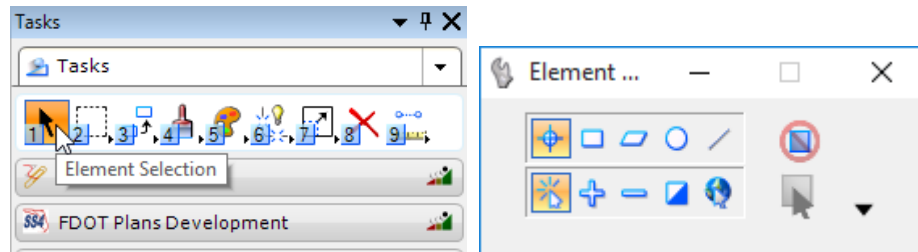
While the Element Selection tool is selected, the pointer becomes an arrowhead with an aperture encircling the tip. The aperture denotes the design plane area in which MicroStation searches for elements. The aperture size or Locate Tolerance is a user preference that is adjustable in the Preferences dialog (Input category).

If any elements are selected, an arrowhead icon is displayed in the MicroStation Status Bar. The number to the right of the icon is the number of selected elements.
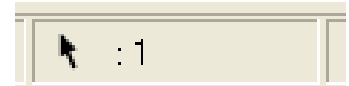
The Element Selection tool is access from the Main task embedded at the top of the Task Dialog located by default on the left side of the MicroStation application window. <OR> The Element Selection tool can be access from the Main Classic task as well.

Note   MicroStation sets the abbreviated Element Selection tool settings dialog as the default to display when a new session is opened.

Expanded, the Element Selection tool settings dialog offers a multitude of options for specific selections tasks in creating a set of selected elements called the *selection set*.
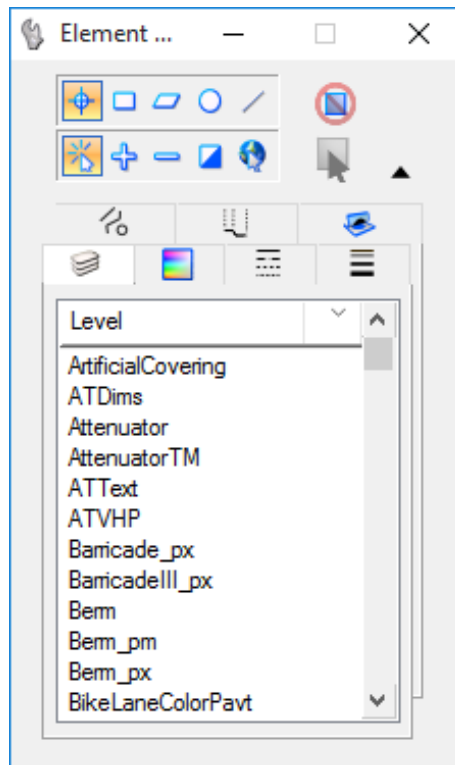
Tip   *If the desired element is visible, but cannot be selected, check the following:*

- *To select a shape, the pointer must be close to one of the enclosing lines. To select a circle or an ellipse, the pointer must be close to the circumference or center. A filled element can be selected by its interior or surface, which is set in the Locate Interiors option menu of the Preferences dialog Input category.*

- *If Level Lock is on, elements can only be selected on the active level.*

ELEMENT SELECTION SETTINGS

The upper portion of the dialog contains two rows of icons that allow the user to choose the *Method* and *Mode* of creating or editing a selection set and to *Disable* or *Select Handles*.

Tip   *Dragging from Left-to-Right to select elements will use the (Inside) option. Dragging from Right-to-Left will use the (Overlap) option.*

Method:

|		
⊕	*Individual*	Allows the user to select individual elements graphically. To select additional elements (when mode is set to New) hold the <Ctl> key, then click on elements.
□	*Block (Inside)*	All elements inside the defined block are selected. Click this icon to toggle between *Inside* and *Overlap*.
⊕	*Block (Overlap)*	All elements inside plus those overlapping the defined block are selected. Click this icon to toggle between *Inside* and *Overlap*.
▱	*Shape (Inside)*	All elements inside the defined shape are selected. Click this icon to toggle between *Inside* and *Overlap*.
⬦	*Shape (Overlap)*	All elements inside plus those overlapping the defined shape are selected. Click this icon to toggle between *Inside* and *Overlap*.
○	*Circle (Inside)*	All elements inside the defined circle are selected. Click this icon to toggle between *Inside* and *Overlap*.
◌	*Circle (Overlap)*	All elements inside plus those overlapping the defined circle are selected. Click this icon to toggle between *Inside* and *Overlap*.
╱	*Line*	Elements are selected by defining a line that intersects them.

Mode:

✳	*New*	Clears the current selection set and starts a new selection set.
✚	*Add*	Adds elements to the selection set.
▬	*Subtract*	Removes elements from the selection set.
◪	*Invert*	Toggles the selection status of an element (selects deselected elements and deselects selected elements).
🌐	*Select All /*	Deselects all selected elements, or selects all elements if no elements are currently selected.
✳	*Clear*	
⊘	**Disable Handles**	If on, handles are not displayed for selected elements.
◹	**Select Handles**	If on, the Block / Shape / Circle / Line selection methods and the Add / Subtract / Invert / Clear selection modes apply to selecting handles rather than elements. (To use this icon, Disable Handles must be off.)

ELEMENT SELECTION EXTENDED SETTINGS

Selecting the Show Extended Settings arrow expands the *Element Selection* window to reveal the *Attribute* tabs. These tabs allow for selection of elements by one or more attributes by clicking on any given attribute in any tab. Alternatively, when selecting elements graphically, the active set of attributes displays as a highlighted group in each tab's list box.

If choosing an attribute that exists in a complex element such as a cell, the entire complex element is selected. If choosing a Color, Line Style, or Line Weight attribute, Element Selection finds all elements with this active symbology, including ByLevel attributes.

*Note*   All tabs are not visible by default. Right-clicking on a tab will display a pop-up menu to customize the tabs displayed.
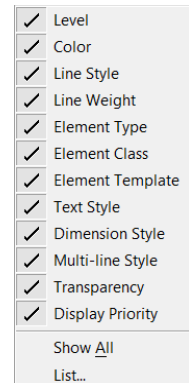
	Allows selection of the required levels from a list of levels in the model displayed in the window below the tab heading.
Level	
Color	Allows selection of the required colors from a list (0–254 plus ByLevel).
Line Style	Allows selection of the required line styles from a list (standard line styles plus custom line styles and ByLevel).
Line Weight	Allows selection of the required line weights from a list (0–31 plus ByLevel).
Dimension Styles	Allows selection of the required dimension style from a list.
Multi-line Styles	Allows selection of the required multi-line style from a list.
Transparency	Allows selection of the required transparency style from a list.
Display Priority	Allows selection of the required display priority from a list.
Element Type	Allows selection of the required element types from a list.
Element Class	Allows selection of the required element classes from a list.
Element Template	Allows selection of the required element template from a list.
Text Styles	Allows selection of the required text style from a list.

SELECTION HANDLES

By default, selected elements are bracketed with squares called *Edit Handles (or handles).* They are used to modify or manipulate elements. They are represented by on-screen interactive graphics that are associated with a particular user action. There are several types of handles that perform different functions.

The above elements can be modified and manipulated by selecting them first with the Element Selection tool, which displays handles for the element, named fence, or reference boundary display. When handles are displayed:

- Drag a handle to modify the element, while maintaining geometric constraints, such as angles at vertices and radius ratios.

- Press the <Alt> key while dragging a single handle, to modify the element without maintaining geometric constraints.

- Drag any other part of the selected element to move or copy it. To copy it press and hold down the <Ctrl> key once dragging starts.

- Use <ctrl-data point > to select multiple handles and modify them simultaneously.

- Press the <Ctrl> key while dragging a selection set around a set of handles to select the handles.

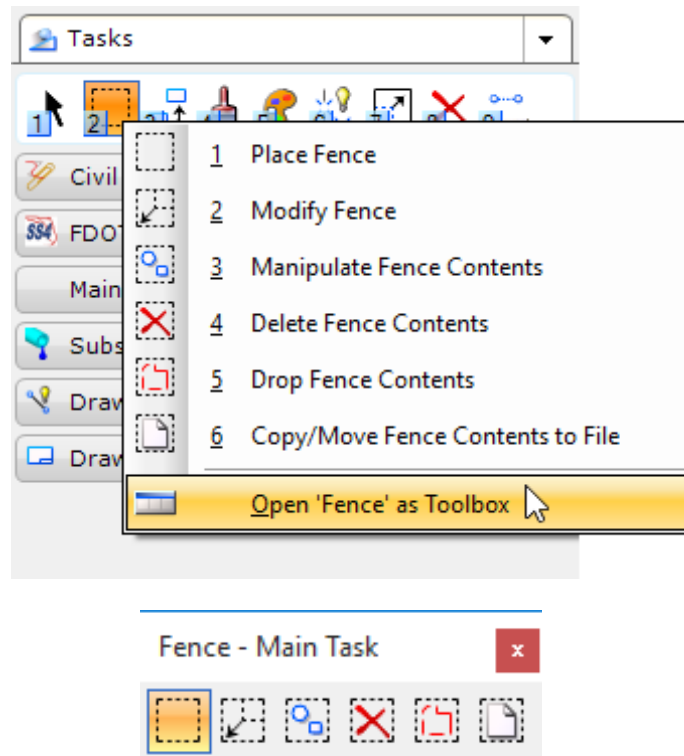- Where multiple elements are selected, select handles on any of the highlighted elements.

USING FENCES

A *fence* is used to create a temporary grouping of objects that ends when the DGN file is closed, if not before. The group is defined by placing a fence object in the design file, just as the pointer is dragged around elements to select them with the *Element Selection* tool. A fence has two additional grouping capabilities:

- As a void to exclude elements inside (or overlapping) the fence boundary.

- To clip elements that cross the fence boundary, like a cookie cutter, so that only the parts inside the fence boundary (or outside in the case of a void) are subject to manipulation.

Fences can be rectangular (fence block), polygonal (fence shape), or circular (fence circle). A fence shape can be derived from a previously placed shape element. Fences are persistent in both 3D and 2D DGN files. Place a fence or zoom in on the design and the fence will remain when zooming back out.

The Fence toolbox can be accessed from the Main Task bar or from the Main Classic Task bar.

	Place Fence	Used to place the fence.
	Modify Fence	Used to move the fence or modify one of its vertices.
	Manipulate Fence Contents	Used to manipulate fence contents or extend/shorten elements that overlap the fence.
	Delete Fence Contents	Used to delete fence contents.
	Drop Fence Contents	Used to break up the complex elements included in the fence contents into their components.
	Copy/Move Fence Contents to File	Used to copy or move the fence contents to a new DGN file.

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PLACE FENCE

The *Place Fence* tool is used to place the fence. On placement, the fence, whether a fence block, shape or circle, is displayed on screen as a closed shape with the color used to highlight identified elements. While the fence is placed, a fence mode icon displays in the Status Bar.
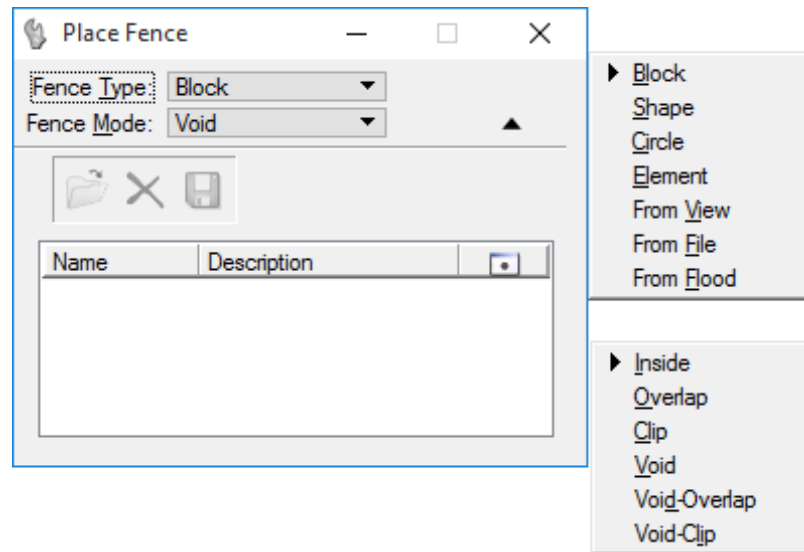
> **Note** It is not necessary to place the fence in order to set the Fence Selection Mode. Conversely, once the fence is placed, the user can adjust the Fence Selection Mode to change the fence contents without moving or replacing the fence.

Fence Type	The Fence Type determines the general shape of the fence and how it is placed.
Block, Shape, or Circle	Used to set the geometric characteristics of a fence for graphical placement.
Element	Uses an existing element to create a fence. The element selected must be a closed shape.
From View	Sets a fence to include the contents of the selected view.
From File	Sets a fence to include the contents of a DGN file. An option menu allow the choice of the following:

- *Active* — elements in the active model from a selected view.

- *All* — elements in the active model and its references, from a selected view.

- *Choose* — elements from a selected reference (or the active model), which are identified by identifying an element from the required model/reference

From Flood	Sets a fence to include the (minimum) area enclosed by a set of elements.

|	
Fence Mode	Sets the Fence (Selection) Mode that, in conjunction with the fence placement, defines the fence contents for manipulation.
Inside	Only those elements completely inside the fence are processed.
Overlap	Only those elements inside or overlapping the fence are processed.
Clip	Only elements completely inside the fence and the parts of elements inside the fence are processed.
Void	Only elements completely outside the fence are processed.
Void-Overlap	Only elements outside or overlapping the fence are processed.
Void-Clip	Only elements completely outside the fence and parts of elements outside and overlapping the fence are processed.
Named fence tools	Clicking the Expand arrow expands the *tool settings* to reveal the Named Fence tools. Double-click in the Name or Description column to modify this information for a named fence. Use the Display column to toggle on and off the display of a named fence.

- *Activate Named Fence* - Makes the named fence (selected in the named fence list) the active fence.

- *Delete Named Fence* - Deletes the named fence selected in the named fence list.

- Create Named Fence from Active Fence - Creates a named fence.

- *Named fence list* - Lists all named fences in the model.

Note   Nearly all of the tools in the Manipulate toolbox have a *Use Fence* option in their *Tool settings*. If this option is selected, and a fence is present in the drawing, the *Manipulate* tool affects the fence contents.

MODIFY FENCE

The *Modify Fence* tool allows the user to move a fence or modify one of its vertices after it is placed. There are two modes available to choose from, **Vertex** or **Position**.

The **Vertex** mode allows the user to select a vertex along the fence perimeter and shift it to a new location. The **Position** mode allows the user to select and move the position of the entire fence.
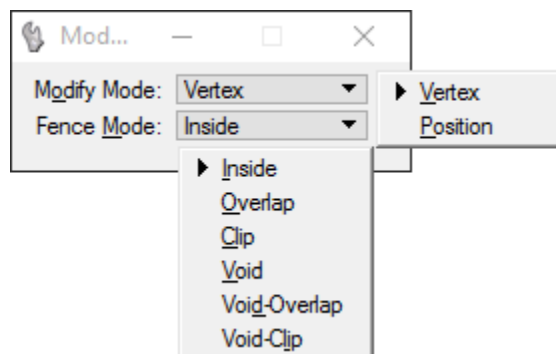
MANIPULATE FENCE CONTENTS

The *Manipulate Fence Contents* tool provides the means to change the contents of the fence based on the provided operations: copy, move, scale, rotate, mirror, array or stretch. These are typical of the options available in the Manipulate toolbox.

The method of manipulation is determined by the tool setting Operation. With the exception of Stretch, each Method works identically to a tool in the Manipulate toolbox with the tool setting *Use Fence* on.
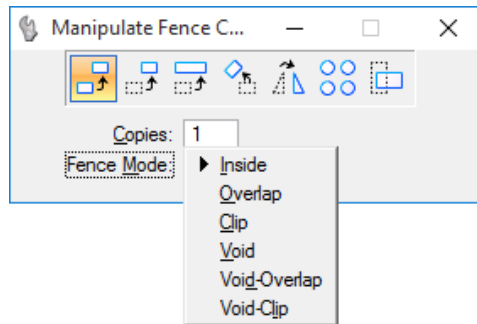

The *Stretch* operation causes the *Manipulate Fence Contents* to ignore the Fence (Selection) Mode and operates on elements as follows:

- A vertex of a line, line string, multi-line, or shape is moved if it is inside the fence. A vertex outside the fence is not moved.

- An endpoint of an arc is moved if it is inside the fence. An arc endpoint outside the fence is not moved.

- An ellipse, circle, or cell is moved if it is completely inside the fence.
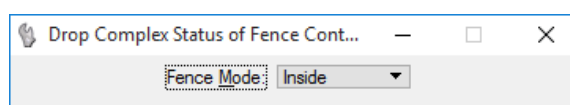
DELETE FENCE CONTENTS

The **Delete Fence Contents** tool can be used as an inclusive grouping of elements or an exclusive grouping with the use of the Inside mode and the Void mode. Elements may also be clipped if overlapping the fence.
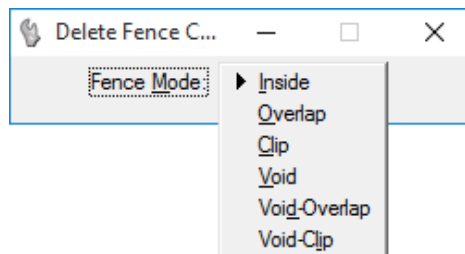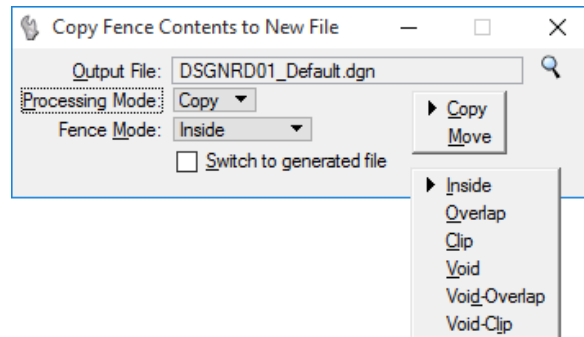

DROP FENCE CONTENTS

Having the same modes for control as **Delete Fence**, the **Drop Fence Contents** tool allows the user to drop complex status of multiple elements. This breaks up the complex elements contained in a fence (cells, complex chains, complex shapes, text nodes, surfaces, and solids) into their components. The elements can then be manipulated individually.

COPY/MOVE FENCE CONTENTS TO FILE

The **Copy/Move Fence Content to File** tool is used to copy or move the contents of the active fence to a new DGN or DWG file.


Output File	Displays the name of the file to which the fence contents will be copied/moved. By default, the file is given a name in the format *filename_model*. Select a different file name by clicking the Define Output File icon (🔍).
Define Output File (icon)	Opens the Save Fence Contents As dialog, which allows for the selection of a destination folder and file name to which the contents will be copied/moved.
	If an existing file is chosen, its current contents will be deleted. An Alert box will warn of this before acceptance of the operation.
Processing Mode	Defines how the fence contents will be processed.
	Copy — Fence contents will be copied to the new file. The original elements will remain in the active model.
	Move — Fence contents will be deleted from the active model and moved to the new file.
Fence Mode	Determines which elements are included in the fence contents.
Switch to generated file	If on, after processing the fence, the current file is closed and the Output File is opened.

SELECT BY ATTRIBUTES

The Element Selection tool allows the user to select information in the drawing by choosing it on the screen. These tools are used to make most of the selections in a drawing. There are times, however, when selections may need to be made from specified criteria instead of a simple graphic selection.

The Select By Attributes tool provides additional attribute-based element selection option: selection by attached tag values and by properties. allows the user to define search criteria and then elements meeting that criteria are selected. This tool also provides the capability to filter the display of elements based on their attributes.

The Select By Attributes dialog is activated through the MicroStation menu option **Edit > Select By Attributes**. When executed, the entire drawing is searched with the criteria selected. This proves to be very useful for selecting items scattered throughout a drawing.
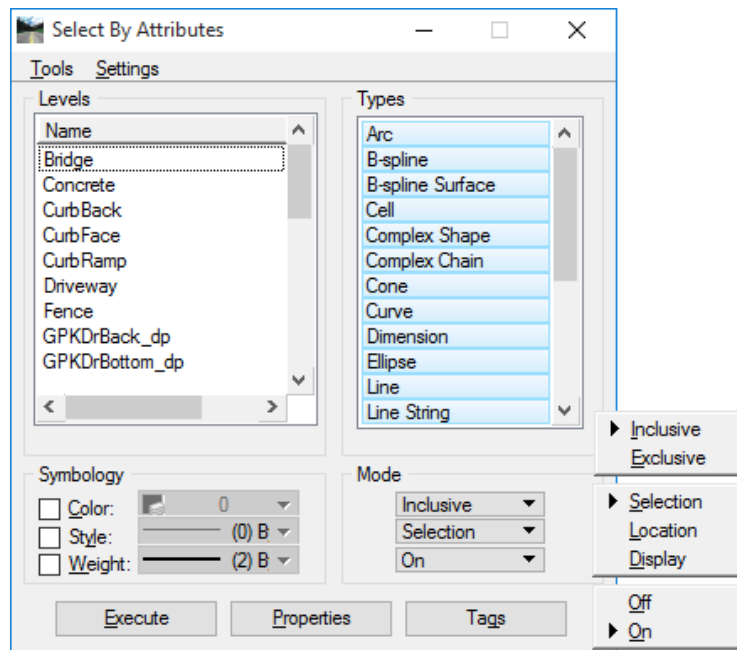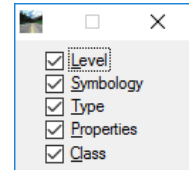
MENU OPTIONS

The Select By Attributes dialog has two pull-down menus in the upper left, *Tools* and *Settings*, which assist in controlling and saving the selection set criteria.
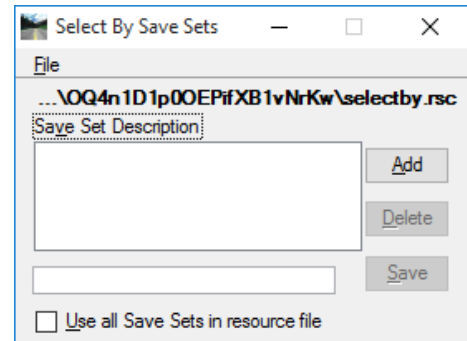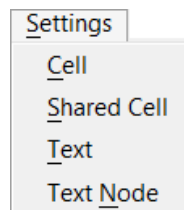
TOOLS
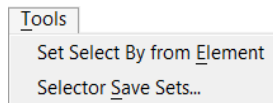
- **Set Select By from Element -** The Set Select By from Element allows the user to set the search criteria to match an element's attributes.

- **Selector Save Sets -** The Select Save Sets allows the user to save and retrieve sets of search criteria settings.
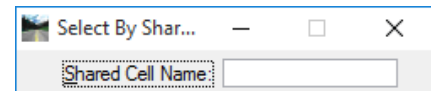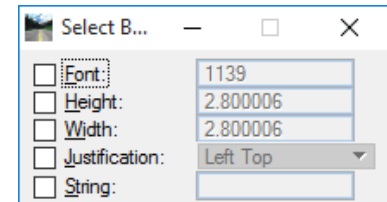
SETTINGS

- **Cell** - Opens the Select By Cell dialog, which is used to specify a criterion based on the name of an unshared cell.

- **Shared Cell** - Opens the Select By Shared Cell dialog, which is used to specify a criterion based on the name of a shared cell.

- **Text** - Opens the Select By Text dialog, which is used to specify criteria based on element attributes specific to text elements. Text strings are case-sensitive. For example, inputting the string "North" would select "North", but not "north", nor "NORTH".

- **Text Node** - Opens the Select By Text Node dialog, which is used to specify criteria based on element attributes specific to text node elements. Text strings are case sensitive. For example, inputting the string "North" would select "North", but not "north", nor "NORTH".
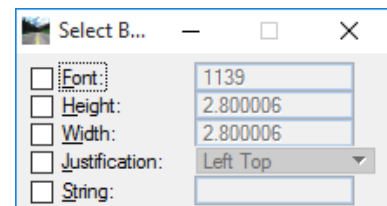
SETTING OPTIONS

Levels   The *Levels* Name list box will populate with the levels used within the design file. These levels can be individually selected or by multiple groups.

- To select a single level, click on the level name.

- To select many levels at once, click and drag through the level list.

- To select non-continuous levels, press <Ctl> and click required levels.

- To deselect a level, press <Ctl> and click the highlighted levels.

Types   The *Types* list box is populated with all recognized MicroStation file types, which can be selected in the same manner as Level names.

Symbology   The *Symbology* section allow the use to specify criteria based on element color, line style (including custom line style), and line weight.

Mode   There are three unlabeled button options that control the following:

- Top Button - Controls which elements are selected, displayed, or located when the Execute button is clicked.

    o Inclusive — All elements that match search criteria.

    o Exclusive — All elements that do not match search criteria.

- Middle Button — Controls how criteria are used when the Execute button is clicked.

    o Selection — Elements are selected based on Select By criteria.

    o Location — Filters location of elements based on Select By criteria. Elements can then be selected by using the Select All command.

    o Display — Filters display of elements based on Select By criteria.

- Bottom Button –

    o If set to On (the default), Select By criteria is effective.

    o If set to Off, Select By criteria is ignored.

Properties   The *Properties* button opens the Select By Properties dialog, which is used to specify criteria based on less prominent element attributes, such as area and class.
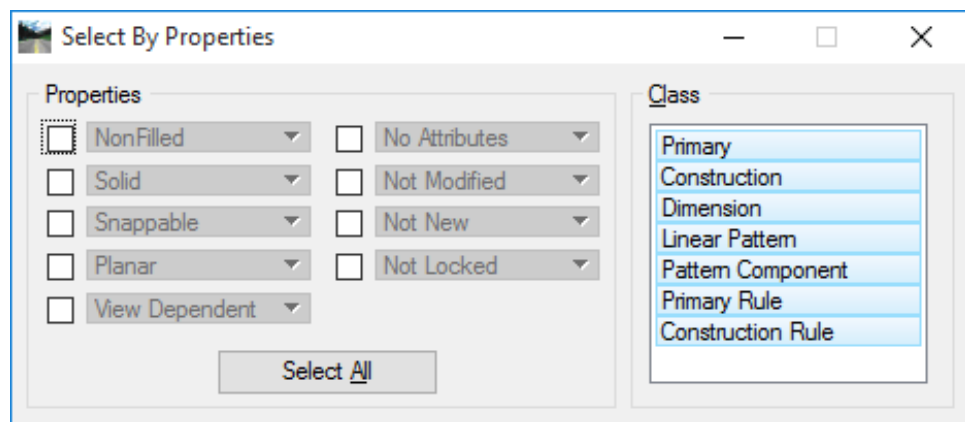
Tags   The **Tags** button opens the Select By Tags dialog, which is used to specify criteria based on tag values. If selection criteria based on tag values are specified, elements that do not have attached tags with the specified tag name(s) will *not* be selected, located, or displayed.
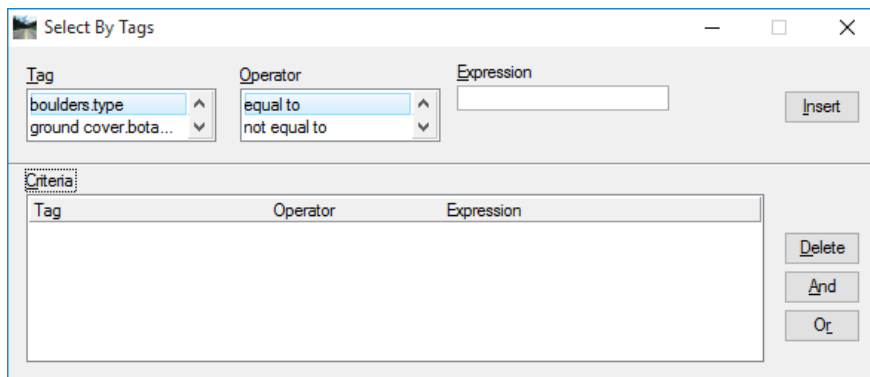
Execute   The Execute button Selects, Locates, or Displays elements in accordance with the specified search criteria and Mode.

EXITING

When Select By Attributes is closed by selecting the [×] icon, an Alert message displays as shown below. The actions taken here determine whether or not selected items are still affected.
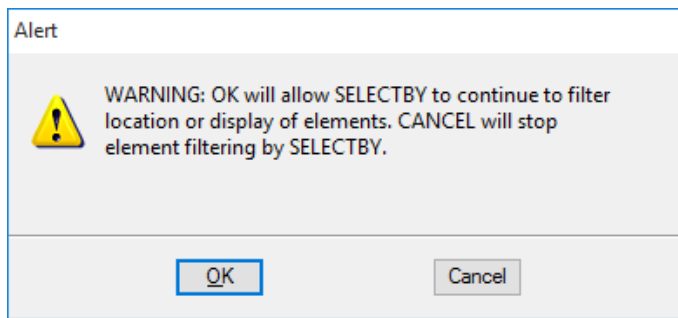
GROUPING ELEMENTS

No matter what method is used to create a Selection Set, the selection is only temporary. MicroStation provides permanent solutions to select a number of elements and grouped them for manipulation as a single element called a *Group*. A Group is a complex element whose component elements need not be connected.

A Group is actually an unnamed cell. Unlike named cells, groups are not defined in cell libraries. For this reason groups are sometimes called "orphan cells." The user creates a Group by selecting the elements and then, from the MicroStation menu, selects **Edit > Group**.

Permanent groupings include: *levels, complex chains and shapes, groups, graphic groups,* and *named groups*.

The tools in the *Groups* toolbox are used to create and manipulate complex chains, complex shapes, and graphic groups. The Groups toolbox can be accessed from the Task Dialog under the Main task.
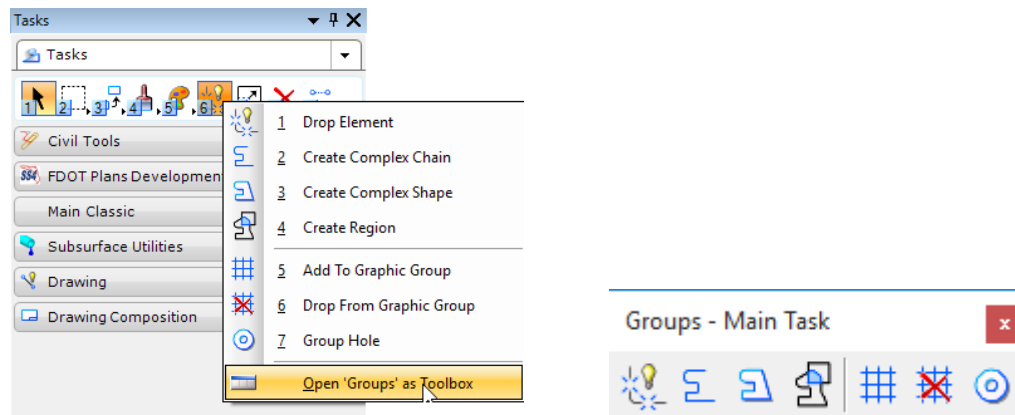
	Drop Element	Break up a complex element(s) or an element(s) of a special type into simpler components.
	Create Complex Chain	Create a complex chain (open complex element).
	Create Complex Shape	Create a complex shape (closed complex element) from individual open elements.
	Create Region	Create a complex shape from the union, intersection, or difference between closed elements or by "flood fill."
	Add to Graphic Group	Create a graphic group. <OR>
		Add elements to an existing graphic group. <OR>
		Combine two or more graphic groups into one graphic group.
	Drop from Graphic Group	Remove (drop) an element(s) from a graphic group. <OR>
		Break up a graphic group into individual elements.
	Group Hole	Select a solid element and the hole or holes to be associated with the solid.

COMPLEX ELEMENTS

Complex Elements are created when a series of primitive elements (lines, line strings, arcs, or curves) are joined together to form a single element. There are many advantages to using complex elements over primitive elements.

For example, if a roadway centerline has been created using lines and arcs, each item would have to be copied parallel individually. This would result in many mouse clicks. If the centerline were made into a complex chain before being copied, a single parallel copy operation would copy the entire centerline with one mouse click.
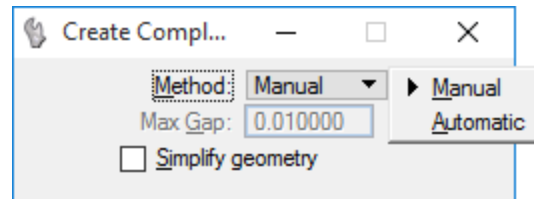
A complex object is always created on the Active Level. The resulting complex chain takes on the active element attributes regardless of the attributes of the component elements.A complex object is an element like any other, and all of the standard manipulation and modification tools apply.

Once a series of primitive objects have been converted to a complex object, they may be broken up in to single objects again with the **Drop Element** tool.

CREATE COMPLEX CHAIN

A *Complex Chain* (sometimes called a "complex string") is a series of connected elements (lines, line strings, arcs, curves) in an open shape and grouped for manipulation as a single entity. The resulting complex chain takes on the active element attributes regardless of the attributes of the component elements.

A *Complex Chain* can be created from the Group toolbox by selecting the ⊆ icon.

Method	Sets how elements are added to the complex chain:

- Manual — Each element is manually selected.

- Automatic — After the first element is selected and accepted, if endpoints of additional open element(s) are within the Max. Gap distance of each other, they are included automatically. Where there is a choice of two or more elements (at the endpoint of an element) then the process allows for the choice to either Accept the highlighted element, or Reset to see the alternative(s).

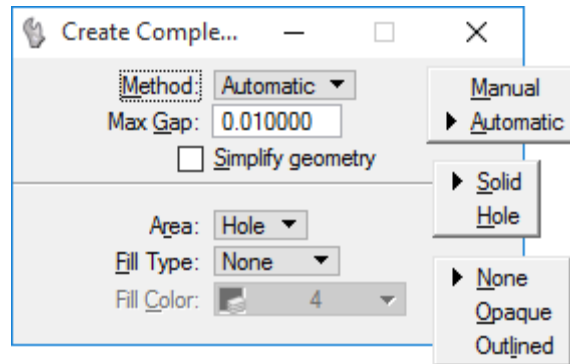|	
Max(imum) Gap	The greatest allowable distance between elements when the Method is Automatic.
	If zero, only elements that connect (have a common endpoint) can be added.
Simplify geometry	If on, connected lines are added as line strings. If selecting only connected lines, the tool produces a primitive line string element rather than a complex chain.

Note	When creating a complex chain, if a fork is found where more than one element is within the *Max Gap* distance, the message "**FORK – Accept or Reset to See Alternate**" displays in the Status Bar.

CREATE COMPLEX SHAPE

A *Complex Shape*, like a complex chain, is a series of connected elements in a closed shape and grouped for manipulation as a single object. The difference between a complex chain and a complex shape is that the first and last element in a complex shape are connected, thereby creating a closed shape.

Although using the *Max Gap* or the *Manual* option fills in gaps between elements when creating complex elements, it is recommended that elements be joined before creating complex elements.

A *Complex Shape* can be created from the Group toolbox by selecting the ⊡ icon.

| **Method**	Sets how elements are added to complex shape.
	• Manual — Each element is manually selected.
	• Automatic — After the first element is selected and accepted, if endpoints of additional open element(s) are within the Max. Gap distance of each other, they are included automatically until a closed shape is created. Where there is a choice of two or more elements (at the endpoint of an element) then the process allows for the choice to either Accept the highlighted element, or Reset to see the alternative(s).
Max(imum) Gap	The largest distance allowed between consecutive elements, if Method is Automatic.
	If zero, only elements that connect (have a common endpoint) can be added.
Simplify geometry	If on, connected lines are added to the boundary as line strings. If selecting only connected lines, the tool produces a primitive shape element rather than a complex shape.
Area	Sets the type of shape created.
	• Solid — The shape created is solid (can be hatched/patterned).
	• Hole — The shape created is a hole (cannot be hatched/patterned).
Fill Type	Sets the active Fill Type.
	• None — No fill
	• Opaque — Filled with Active Color
	• Outlined — Filled with Fill Color (outline of shape takes the Active Color)
Fill Color	Complex shape is filled with this color and optional gradient if the Fill Type is Outlined; otherwise disabled (dimmed).

Note   I If no element is found within the Max Gap, a line element is created between the open endpoints of the first and last elements and the complex shape is closed.

CREATE REGION

The *Create Region* tool has many options and is a more complex tool than the Create Complex Chain or Create Complex Shape. The concepts behind some of its settings have to be understood before this tool can be used.

Using the 🔧 icon from the Group toolbox, *Regions* can be formed from:

- o   The union, intersection, or difference between two or more closed elements.

- o   A region bounded by elements that have endpoints that are closer together than the Max(imum) Gap.
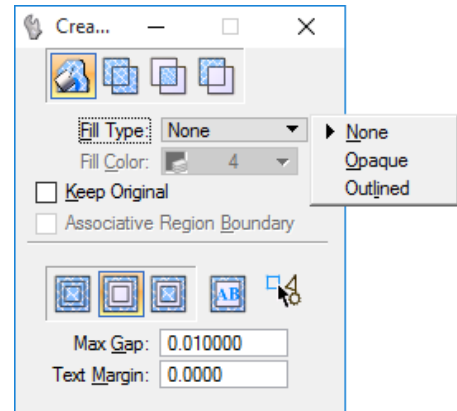
| *Note* | Create Region also uses a Flood option which is the most common way to create regions and is also the easiest to use. |

🪣	**Flood**	Region edges bound the area enclosed by elements that either touch one another or whose endpoints fall within the maximum gap value.
⬜	**Union**	Region edges bound the union of two or more closed, planar elements.
⬜	**Intersection**	Region edges bound the intersection of two or more closed, planar elements.
⬜	**Difference**	Region edges bound the difference of two or more closed, planar elements.

Fill Type	Sets the Active Fill Type. *None* – The complex shape is not filled. *Opaque* – The complex shape is filled with the Active Color. *Outlined* – The complex shape is filled with the Fill Color.
Fill Color	(Fill Type: Opaque or Outlined only) When Fill Type is *Outlined,* Complex shape is filled with this color. When Fill Type is *Opaque*, Fill Color displays the Active Color. When Fill Type is set to **None**, this option is disabled (dimmed).
Keep Original	If on, the original elements remain in the design.
Associative Region Boundary	If on, the region boundary retains association with the elements used to create it. If an original element is modified, then the boundary element updates to reflect the modification.

Flood Region Options

🔲	**Ignore Interior Shapes**	If selected, interior shapes are ignored when the region boundary is calculated.
🔲	**Locate Interior Shapes**	If selected, closed elements inside the selected area are included as part of the new complex shape.
🔲	**Identify Alternating Interior Shapes**	If selected, alternating areas are flooded where shapes are nested inside one another.
🔲	**Locate Interior Text**	If toggled on, any text or dimension text inside or overlapping the selected area is avoided.
🔲	**Dynamic Area Locate**	If toggled on, the area to be included in the region displays dynamically as the user moves the cursor over the view.

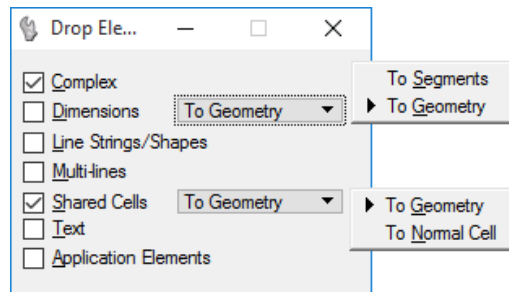|	
Max Gap	Sets the largest distance allowed between consecutive elements. If set to zero, only elements that connect can be added.
Text Margin	Sets the margin to be left around any text or dimensional text included in the selected area.

DROP ELEMENT

The *Drop Element* tool facilitates dropping and reverting a complex element back to its original primitive elements. It is useful when a complex element is no longer desired or cells and other elements need to be broken.

Only the selected elements of the specified type(s) in the *Tool settings* dialog are dropped. The tool has no effect on selected elements of other types. If none of the selected elements is of the specified type(s), the message "Nothing to drop" displays in the status bar.

Complex	If on, complex elements (cell, complex chain, complex shape, text node, surface, or solid) are dropped into their components.
Dimensions	If on, dimension element(s) are dropped into lines, line strings, ellipses, arcs, and text.

- To Geometry — Drops down lines, line strings, ellipses, arcs, and text.

- To Segment — Drops down individual dimension segments.

Line Strings/Shapes	If on, line strings and shapes are converted to series of individual line elements.
Multi-lines	If on, multi-line elements are converted to sets of line strings, lines, and/or arcs.
Shared Cells	If on, the associated option menu allows for the dropping of shared cells:

- *To Geometry* — shared cells are dropped into their components.

- *To Normal Cell* — shared cells are converted to unshared cells.

Solids	(3D only) If on, the associated option menu allows for the dropping of SmartSurfaces or SmartSolids:

- *To Surfaces* — SmartSurfaces or SmartSolids are dropped to simple surfaces.

- *To Wireframe* — SmartSurfaces or SmartSolids are dropped to wireframe elements.

|	
Text	If on, text characters in text elements are converted to the individual elements that are used to draw the characters — lines, line strings, arcs, ellipses, and shapes.
Application Elements	If on, used to drop light weight geometry.

Note	Drop Element cannot be used to drop elements more than one level at a time. For example, if Complex and Line Strings/Shapes are enabled and complex shape is selected that contains two line strings, the complex shape is dropped but the component line strings are not. That is, the operation results in two line strings.

### Exercise 9.1 *Complex Elements*

➢ **Drop Complex Element**

1. Open the MicroStation file c:\e\projects\12345678901\utils*utexrd01.dgn*.

2. Zoom into the cable TV line east of **Station 232+00** and select, as shown highlighted below.

Tip   *The* MicroStation *menu option:* ***Edit > Find/Replace Text*** *can be used to find the station just by entering 232 in the Find field and then click the Find button.*
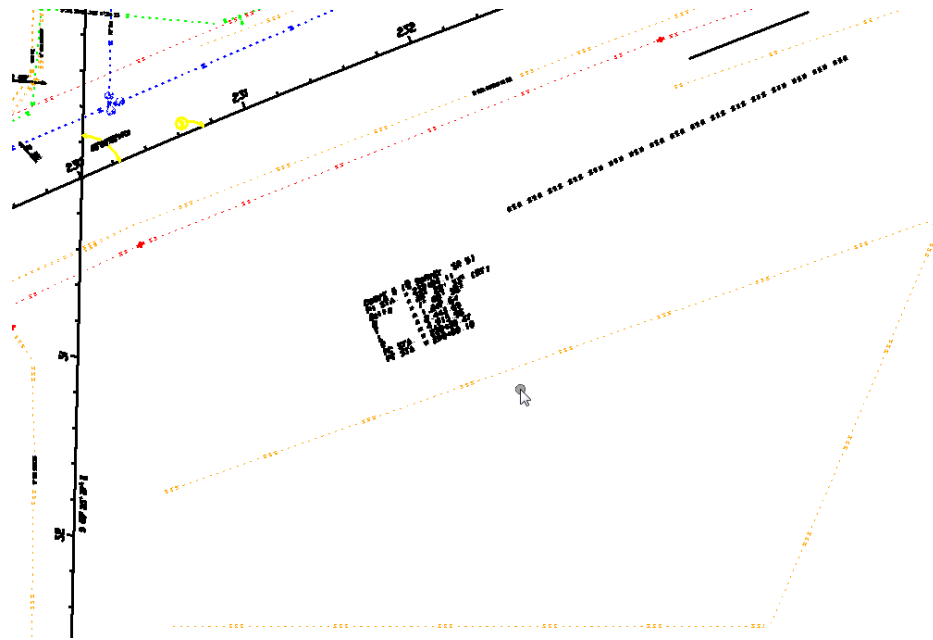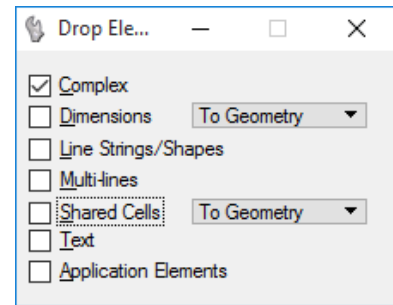

3. Leave the *Active Element Attributes* as they are.

4. From the Main Task, select the **Drop Element** tool from the *Groups* toolbox.
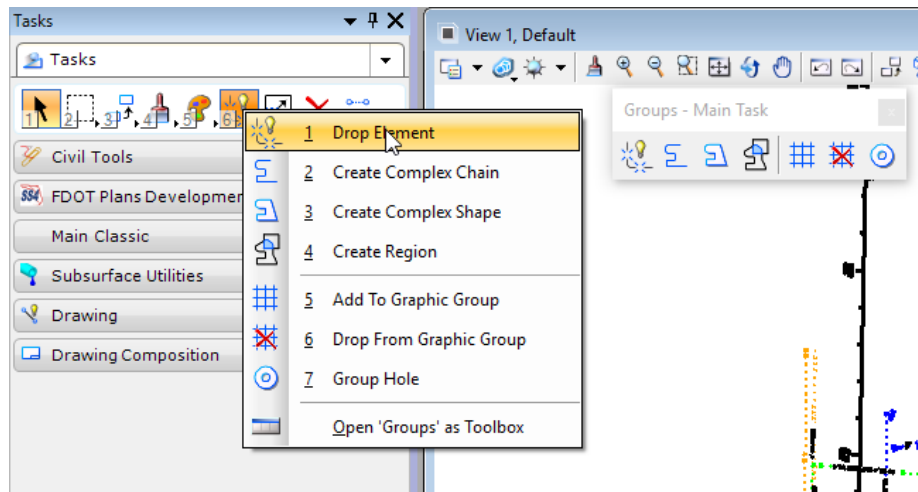
Tip   *If needed, remember to follow the reminder messages in the Status Bar located in the lower left side of the application window.*

5. Set the *Drop Element* Tool Settings as shown.

6. Place a **data point** anywhere on the *cable TV line* as shown below.
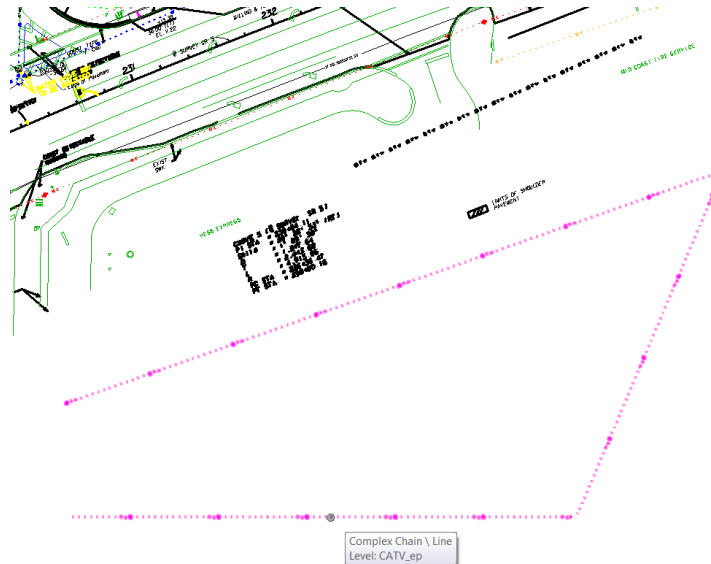
7. Move the cursor over the *cable TV line* and notice that it is now comprised of individual lines and arcs.

➢ *To Manually Create a Complex Chain*

1. Continuing in the same file and location from the previous exercise, set the *Active Level* to **CATVBurB_ep** from the Attributes toolbox at the top of the application window.
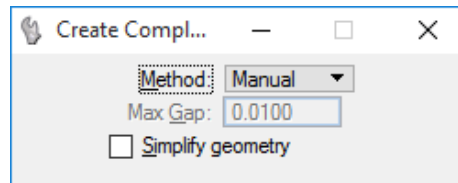
Note Remember: when a Complex Chain is created, it automatically sets to the Active Element Attribute settings regardless of the individual settings of each element selected.
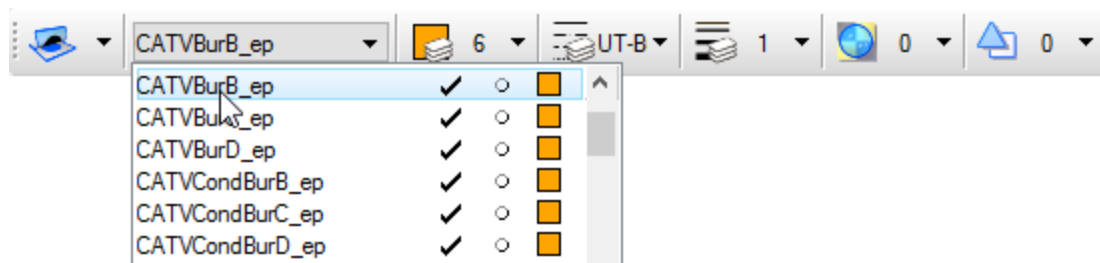
2. From the Main task, select the **Create Complex Chain** tool from the *Groups* toolbox.
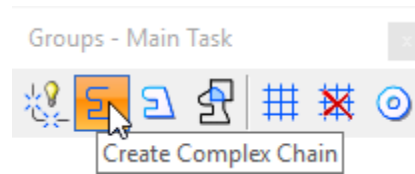
3. Match the *Create Complex Chain* Tool Settings as shown.

4. Place a **data point** on each *cable TV line* elements to select and include in the *complex chain*, as shown highlighted below.

Tip   *In a complex chain, all elements must be connected. If they are not already connected, the elements are connected as they are selected.*
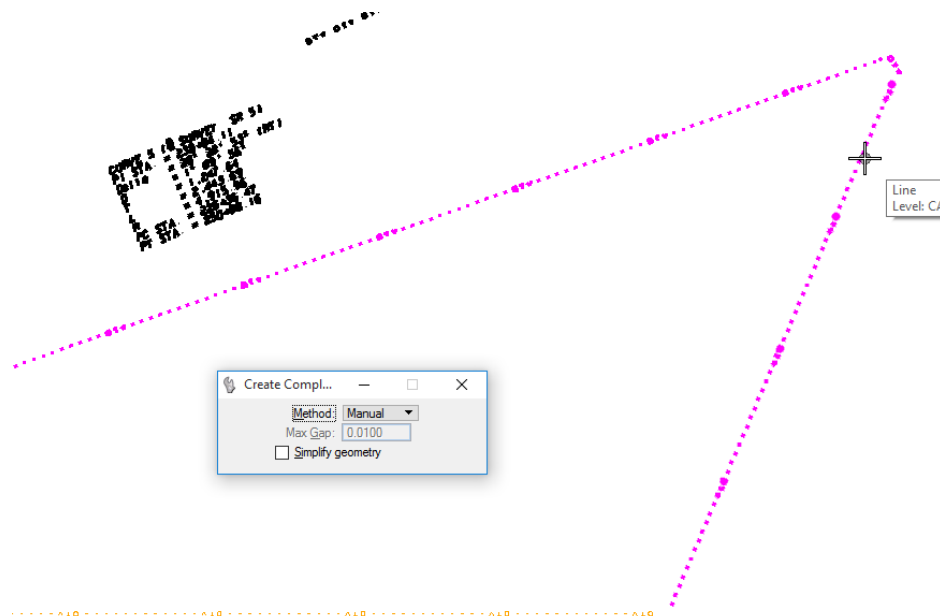
5. When all cable TV line segments are selected, place a **data point** outside the selection set and then click **Reset** (right click) on the mouse to complete the complex chain.

6. Using the *Element Selection* tool, select an element from the *Complex Chain* created. Notice now that the complex element can be manipulated as a whole instead of individual elements.

7. Close the *Create Complex Chain* Tool Settings and, if open, close the *Groups* toolbox.

➢ *To Manually Create a Complex Shape*

The goal is to create a driveway shape similar to the example displayed.

1. Open the MicroStation file c:\e\projects\12345678901\roadway*dsgnrd01.dgn*.

2. Zoom in the driveway locations near **Stations 286+00 – 287+00**.

Tip *The* MicroStation *menu option:*
Edit > Find/Replace Text can be used
to find the station just by entering 286
*in the Find field and then click the Find*
button.
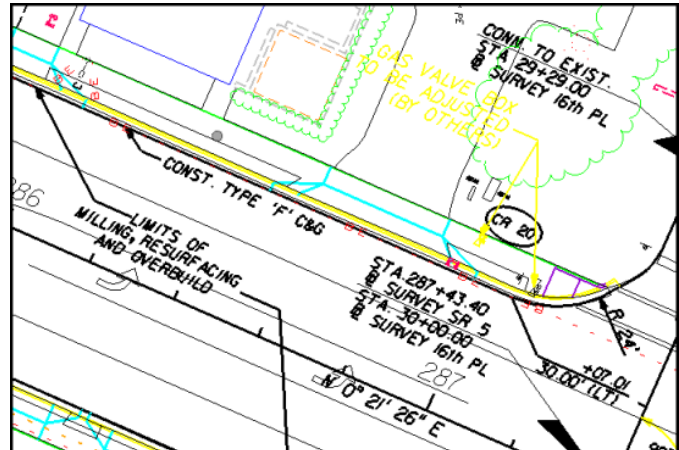
3. To improve the accuracy of selection, turn off surrounding levels that make it difficult to access the driveway elements.

Note   This is for display purpose only and in no way effects the elements within the design file.

   a. Right-click on any element and select "Turn Level Off By Element" in the pop-up context menu.
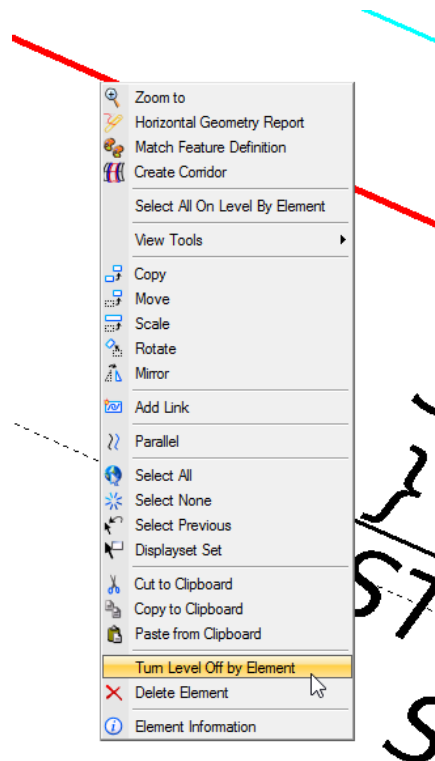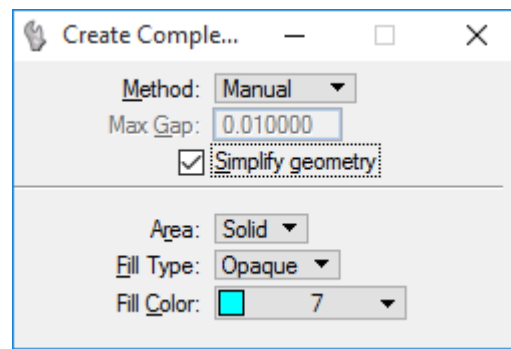
4. From the Attributes toolbox located at the top of the application window, set the *Active Level* to **Driveway**.
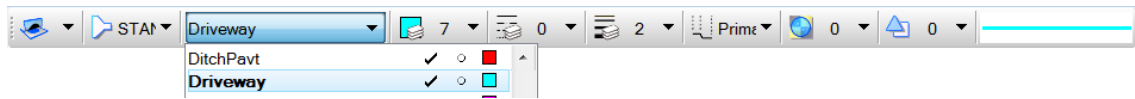
5. Select the **Create Complex Shape** tool from the *Groups* toolbox.
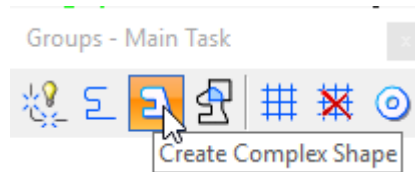
6. From the *Create Complex Shape* Tool Settings, set options as shown.

7. Select each of the **Driveway** elements shown below, that make up the boundary of the shape to be created.

Tip *Select Driveway elements in succession to achieve the shape required. The Complex Shape tool will automatically connect as they are selected. These connections appear as "lines" that connect the endpoints of the elements. They are part of the complex chain but are not elements. They disappear when complex status is dropped. It is recommended that elements be connected before being included in a complex chain.*

8. When all selected elements are highlighted as shown below, place a **data point** outside the selection set.
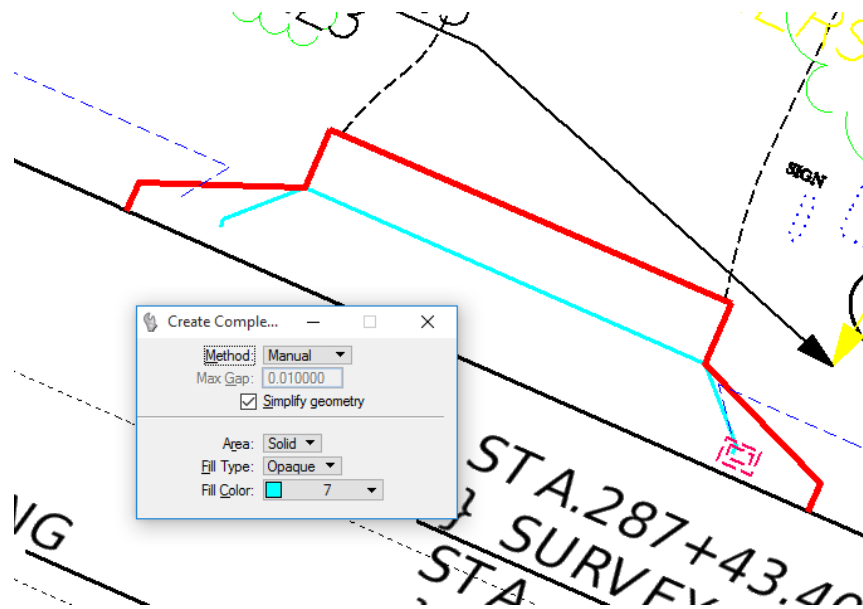
9. **Reset** (right click) on the mouse to complete the complex shape. The shape will automatically fill.
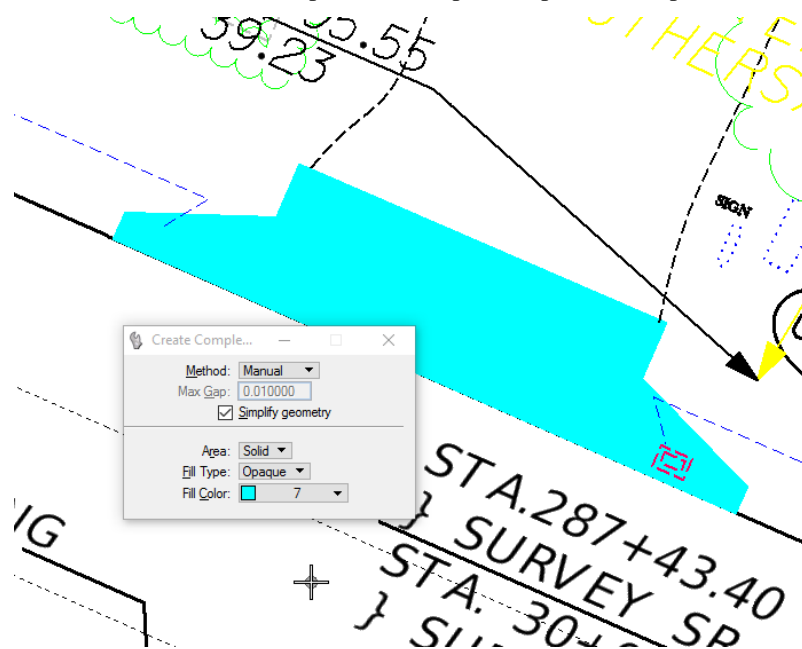

10. Perform the same steps on a few of the other driveways.

11. Exit MicroStation.

GRAPHIC GROUPS

A Graphic Group is a quick method for grouping elements in MicroStation. A design file can contain an unlimited number of graphic groups. The groups only exist within their host design file and cannot be exported to other design files. An element can be a member of only one graphic group at a time. The *Groups* toolbox has tools for creating a graphic group and adding and removing elements to or from a graphic group.

Without "breaking up" the grouping, individual member elements can be added, removed or manipulated. Whether certain actions change the whole Graphic Group or only part of it depends upon whether Graphic Group Lock is turned "on", all elements in the graphic group are manipulated, or "off", only the selected element is manipulated.

An element can be added to a Graphic from the Group toolbox by selecting the ⊞ icon.
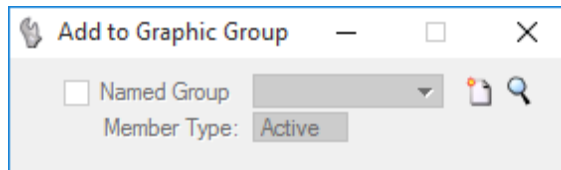
Named Group	(Existing named groups present in model only) If on, selection can be made from the drop-down list of existing named groups present in the model.
Create New Named Group icon	Opens the Create Named Group dialog, which allows for the creation of a new named group.
Open Named Group Dialog icon	Opens the Named Groups dialog, which allows for the creation and modification of Named Groups and their hierarchies.
Member Type	Defines the way that other members of a named group are affected when a member of the named group is manipulated or selected.

An element can be added to a Graphic from the Group toolbox by selecting the ✳ icon.
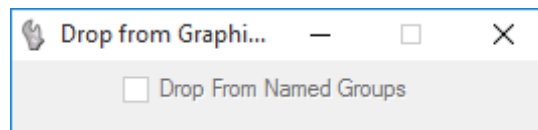
Drop From Named Groups	If on, drops elements from named groups. If off, elements can be dropped from graphic groups only.

NAMED GROUPS

For greater flexibility still, *Named Groups* can be created, which allow a name to be given to each grouping of elements. Named groups can include elements from the active file and those from directly attached references.

A new Named Group can be created from the Groups toolbox using the *Add to Graphic Group* tool <OR> from the MicroStation menu option **Utilities > Named Groups**.
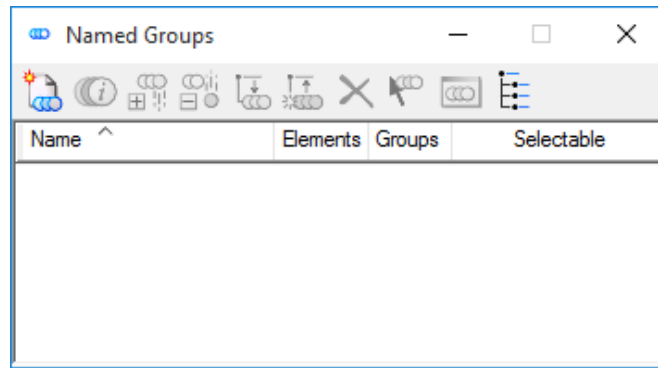
New Named Group	Creates a named group in the dialog. The new group has a default name that the user may change.	
Show Named Group	Displays the Element Properties for all elements in the selected graphic group.	
Add Elements	Allows the user to add elements to the selected graphic group.	
Remove Elements	Allows the user to remove elements from the selected graphic.	
Add Named Group to Parent	Forces the selected graphic group to become a child of another graphic group.	
Remove Group from Parent	Removes the graphic group from being a child of another graphic group.	
Delete Group	Delete the graphic group name from the Named Groups dialog. This does not delete the elements in the graphic group.	
Select Elements in Named Group	Selects all elements in the graphic group.	
Put Elements into Display Set	Sets the contents of the selected graphic group as the Display Set.	
Show Hierarchy	Shows the hierarchy window.	

GROUP HOLES

The Group Holes tool is used to select a solid element and the hole elements to be associated with the solid. The solid and its associated hole(s) can be shapes, ellipses, and/or complex shapes that are in the same plane. Holes are not patterned and appear "transparent" in rendered views.

- A *Group Hole* can be created from the Group toolbox by selecting the ⊚ icon.

ASSOCIATE ELEMENTS

Under most circumstances elements are static, and when an element is placed in a design, its position is defined simply by the design plane coordinates on which it lies. It retains that position in the design plane until moved with an element manipulation tool.

One exception to this rule is association, in which an element's position in the design plane is defined in relation to another element. When that other element is moved, the associated element moves with it. For example, dimension elements can be associated with the elements whose dimensions they display. These dimensions update when the dimensions of the elements, with which they are associated, change.

The types of elements that can be associated with other elements are dimensions, multi-lines, normal cells, shared cells, and tags. The association is made when the dimension, multi-line, cell, or tag is placed.

Exercise 9.2 *Creating Groups*

➢ *To Create a (UnNamed) Group*

1. Open MicroStation file: c:\e\projects\12345678901\utils*utexrd01.dgn*.

2. Use the *MicroStation* menu option: **Edit > Find/Replace Text** to move to the intersection at **Station 257+00** by entering **257** into the *Find* field and then clicking the **Find** button. Close Find/Replace text dialog.
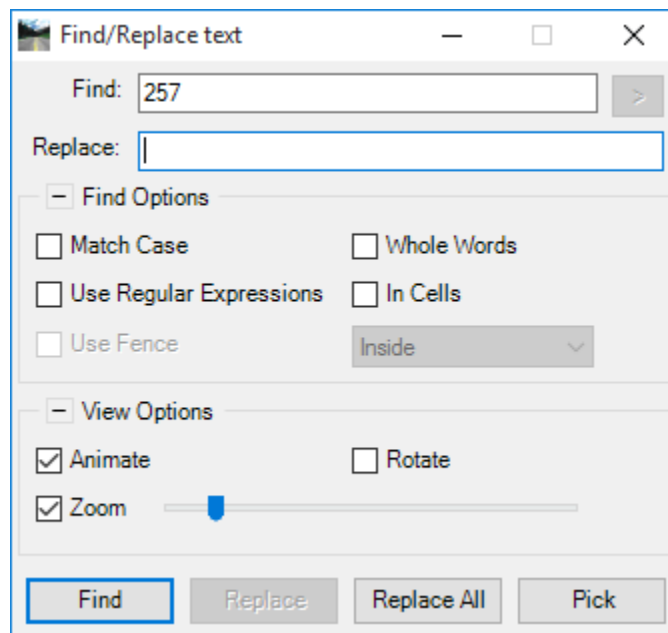
3. From the Main task, select the **Element Selection** tool and open the **Level** tab. Scroll down to select the *Level* **TeleBur_ep** and notice that all elements on this level are highlighted.
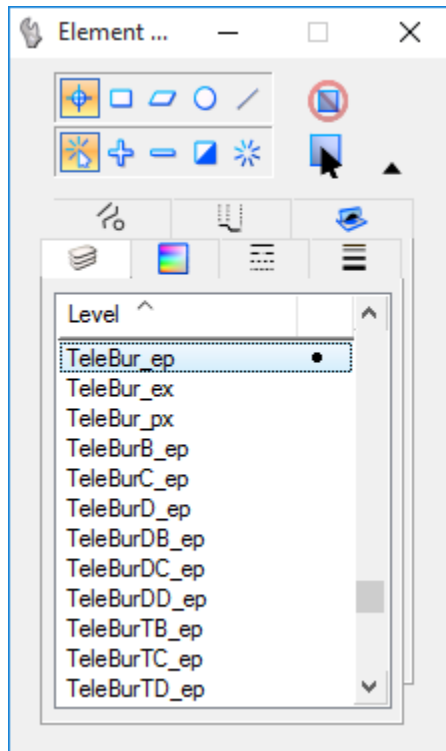

4. From the MicroStation menu, select **Edit > Group**.

5. Hover the cursor over one of the elements from the selection set and notice that all elements in the group highlight at the same time.

6. Select one of the elements of the group and drag to the right. The group will move as one element.

7. From the MicroStation menu, select **Edit > Undo Drag Selection** to set the group back to previous position.

➢ *To Break Up a Group*

1. Select the newly created Group from the previous exercise.

2. From the MicroStation menu, select **Edit > Ungroup.**

3. Move the cursor over the elements and notice they are no longer working as one element.

➢ ***To Create a Named Group with the Add to Graphic Group tool***
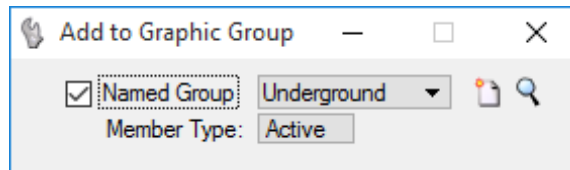
1. Continuing with the MicroStation file: c:\e\Projects\12345678901\utils*utexrd01.dgn*.

2. Leave the *Active Element Attributes* as set.

3. From the *Group* toolbox and select the **Add to Graphic Group** tool.
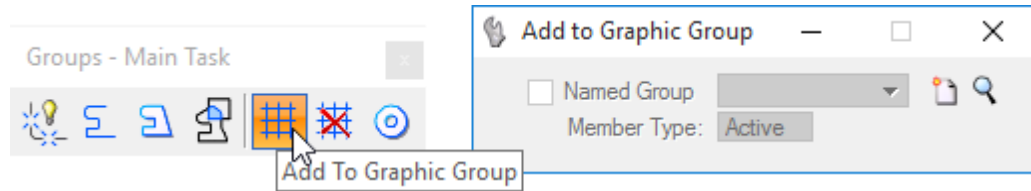
4. From *Add to Graphic Group* Tool Settings, click the **Create New Named Group** icon and key in **Underground** for the new *Name*.

    a. **(Optional)** Key in a *Description* for the new Named Group.

    b. **(Optional)** Select the c*heck* box to *Select all members when any member selected*. This turns on Selectable for the Named Group.
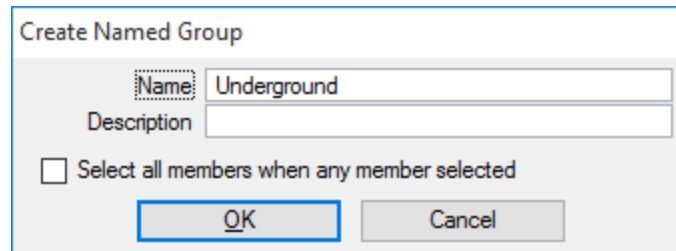
5. Click **OK**. The *Add to Graphic Group* Tool Settings populates with the newly created Named Group.


6. Close *Add to Graphic Group* Tool Settings and the *Groups* toolbox.

➢ *To Create a Named Group with the Named Groups dialog*

1. Continuing with the MicroStation file: c:\e\Projects\12345678901\utils*utexrd01.dgn*.

2. From the *Primary Tool* toolbox located at the top of the application window, select the **Key-in** tool.
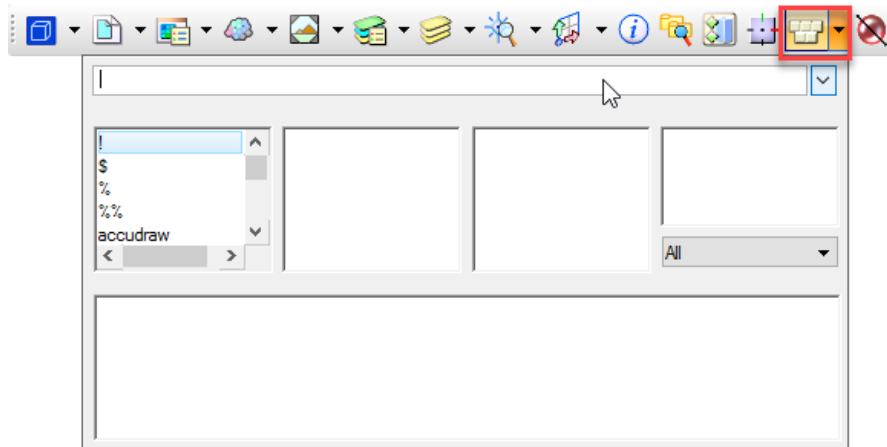

3. Key-in **REFERENCE DISPLAY OFF ALL** and press <**Enter>** to accept. Close the Key-in dialog.
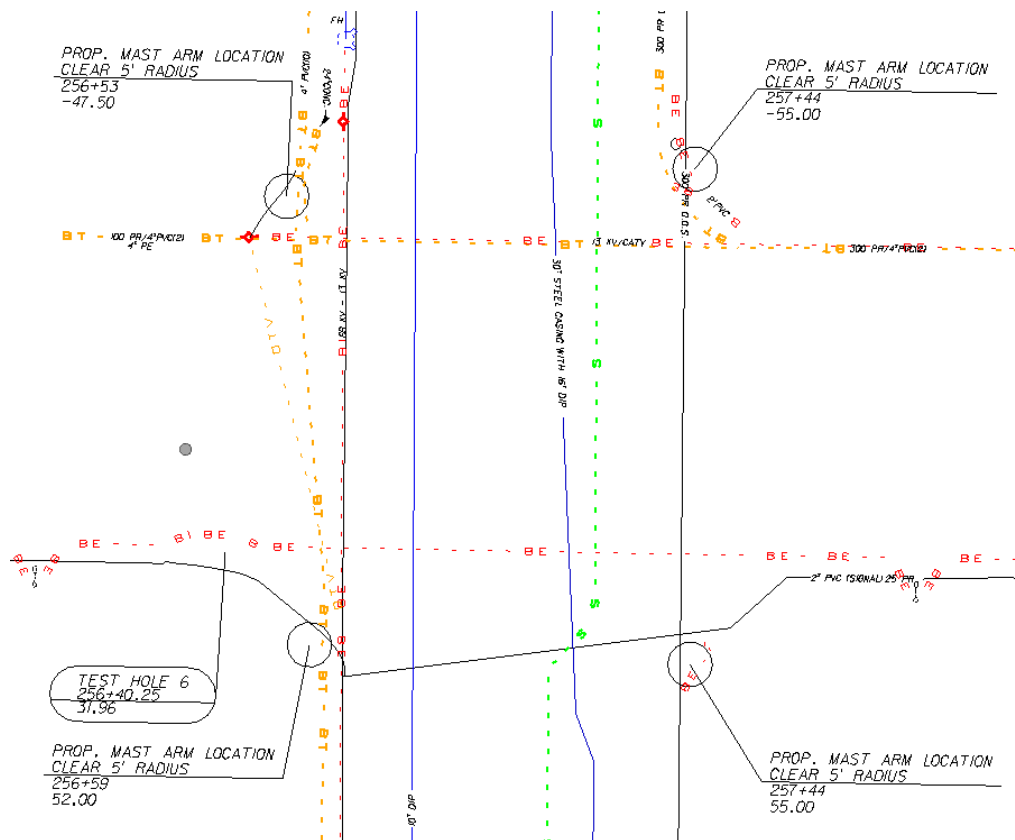
4. Select the elements to be grouped: from the Main task, open the *Element Selection* tool and extend to the *Level* tab. Select *Level* **CATV_ep**. All elements on this level automatically highlight.

Note   Notice that the OTV line left of the intersection is highlighted. Also notice in the lower right on the Status Bar an arrow head appears with the number 9, indicating elements were selected.
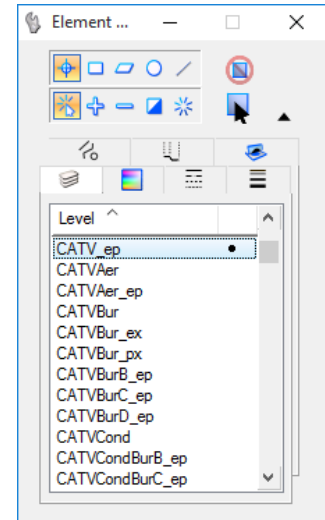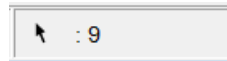
5. Make NO changes to the *Active Element Attributes*.

6. From the MicroStation menu bar, select **Utilities > Named Groups**. The Named Groups dialog opens. Note the previously created Named Group Underground is displayed.
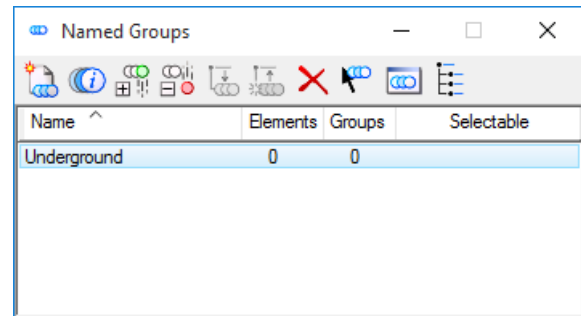
7. From the Named Groups dialog, select the **New Named Group** icon. A new entry is inserted in the list box, with a default name highlighted ready to edit.
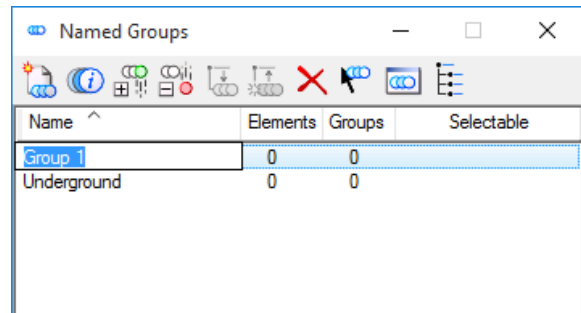
8. Key in **Television** to replace the default name and press **<Enter>** to accept.
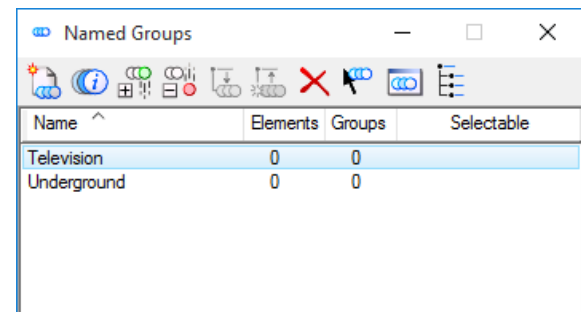
9. Select the **Add Elements** icon. The *Add to Graphic Group* tool is activated, with Named Group turned on, the new named group selected, and Member Type set to Active.
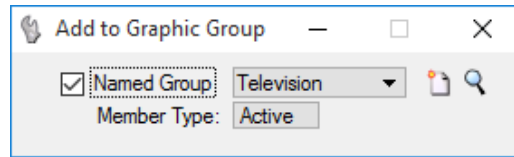

Note   MicroStation message in the lower left status bar: Add To Named Group > Accept to add selected elements to group

10. Using the *Element Selection* tool, accept the selection set by placing a data point on the highlighted OTV line left of the intersection. Notice the Named Groups dialog updates Television with 9 elements matching the count indicated in the Status Bar.
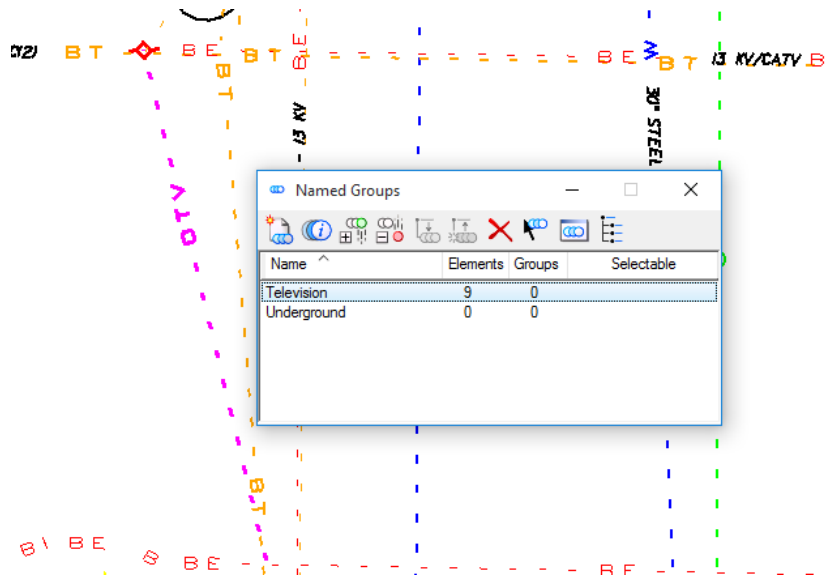
11. Repeat steps 8 and 9 to create the new Named Groups as shown below and leave open.
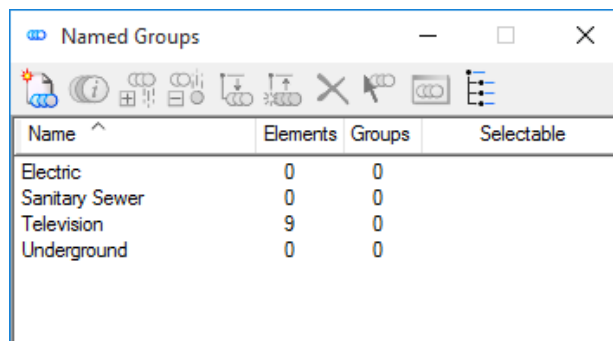
12. Minimize the extended portion of the Element Selection tool settings.

13. Exit MicroStation